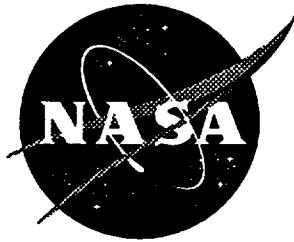


NASA Contractor Report 201582

1N-61
66837



SMART Structures User's Guide - Version 3.0

Jan L. Spangler

Analytical Services and Materials, Inc., Hampton, Virginia

Contract NAS1-20255

June 1996

National Aeronautics and
Space Administration
Langley Research Center
Hampton, Virginia 23681-0001

Table of Contents

Preface 1

...About the Software 1

...About the Guide 1

...Convention 2

Chapter 1 Overview 3

...Who is SMART Structures for? 3

...What kind of capabilities does SMART Structures provide? 3

...How does SMART Structures interface with other programs? 4

...How Does SMART Structures interface with other SMART modules? 4

...Using SMART to generate a structural model 5

Chapter 2 S-WING 13

...Wing Box 14

...Ribs 16

...Spars 18

...Surfaces 19

...Library 22

Chapter 3 S-FUSE 23

...Stations 25

...Breaks 28

...Surfaces 32

...Library 35

Chapter 4 MERGE 37

...The Merge Options 39

...Resetting the Tolerance 42

Appendix A Bringing an Outside Model into S-WING 43

...Bringing in the Surface Model 43

...Selecting from the Tree 44

...Changing the Orientation 44

...Moving the Root Rib Outside the Fuselage 45

...Editing the Planform 45

...*aerowing* is Added to the Tree 47

Appendix B Constraints on Geometric Input to SFUSE 48

...Fuselage 48

...Root Rib 49

...Tank 50

Appendix C Tutorial 52

...Before You Begin 52

...Lesson 1 52

...Lesson 2 68

List of Figures

Illustration of model generated by SMART Structures.	3
Data flow relative to SMART Structures.	5
Process for modeling the structural wing.	6
Streamwise chord-lines mark assembly divisions.	6
Wing planform and internal structure.	7
Some options for defining rib and spar structure.	7
Modeling process for structural fuselage.	8
Stations are placed perpendicular to the axis of the fuselage.	8
Examples of ring frame and bulkhead structure.	9
Typical placement of longeron and beams.	9
Methods for joining spar and ring frame structure.	10
Side-view of wing root rib and fuselage body.	10
Inside the planform editor, the root rib is moved outwards from the center line.	11
The vertical tail is an example of a rotated wing.	11
Illustration of tank generated by SMART Structures	12
Planform and wing box of one wing section.	14
Menus for defining the wing box.	14
The Edge menu and the Active Section slider bar specify which edge to edit.	15
Menus for ribs.	16
Together, the Box menu and the Active Section slider bar determine which part of the wing you are editing.	17
Examples of <i>Equally spaced</i> and <i>Perpendicular to span</i> .	17
Menus for spars.	18
Examples of spar options.	19
Menus for generating the structural surface.	20
Tree that is created for each assembly.	20
Unnecessary surface is generated in X-Z plane for vertical tails.	21
Menus for writing or reading the structural wing to a file.	22
Fuselage display.	24
Menus for defining stations.	26
Both models have the same number of frames (3) but different intervals-between-stations specifications.	27
Labels display top depth, bottom depth and number of intervals-between-stations.	28
Illustration of screen for defining breaks.	29
Beam with the patch-precision set to 3.	30
Menus for generating the structural fuselage.	32
Typical component hierarchy.	32
Vertical drop option for wing carry-through.	33

The <i>combine</i> Option is applied to “Assembly-1”, “Assembly-2”, and “Framewebs” nodes.	34
Illustration of rib added to carry-through structure.	35
Menus for writing or reading the structural fuselage to a file.	35
Unmatched vertices typically occur at the juncture of two fuselage or wing assemblies.	37
Menus for MERGE Mode.	39
Illustration of the “merge” Option.	40
Illustration of the “split” Option.	40
Illustration of the “join” Option.	41
Illustration of the “equivalence” Option.	41
Illustration of the “outline” Option.	42
Example of surface model for wing (left) and for vertical tail (right.)	43
Path for bringing outside model into structural wing editor.	44
Slider bars for editing planform.	46
Menu panel for editing planform.	47
In general, a reflected surface represents the fuselage body.	48
Example of bad input geometry.	49
Constraints on root rib’s geometry.	49
Root rib for vertical tail.	50
Illustration of tree showing position of root rib component of wing body section.	50
Modeling approach for lesson 1.	53
Illustration of screen after step 6 of lesson 1.	54
Illustration of screen after step 9 of lesson 1.	54
Illustration of screen after step 12 of lesson 1.	55
Illustration of screen after step 13 of lesson 1.	55
Illustration of screen after step 14 of lesson 1.	56
Illustration of screen after step 15 of lesson 1.	56
Illustration of screen after step 16 of lesson 1.	57
Structural model for main wing.	57
Illustration of screen after step 19 of lesson 1.	58
Illustration of screen after step 20 of lesson 1.	59
Structural model of vertical tail after rotation.	59
Illustration of tree for vertical tail. Delete nodes shown in reverse video to get rid of unnecessary surface in X-Z plane.	60
Illustration of screen after step 27 of lesson 1.	61
Illustration of screen after step 28 of lesson 1.	62
Illustration of screen after step 29 of lesson 1.	62
Illustration of screen after step 30 of lesson 1.	62
Illustration of screen when <i>display labels</i> is turned on.	63
Illustration of screen after step 34 of lesson 1.	64
Illustration of screen after step 35 of lesson 1.	64
Step 37 of lesson 1 adds a floor and a keel beam (this figure does not show patch precision).	65
Diagram showing how many patches to distribute between breaks.	65

Hierarchy of components for fuselage of *vehicle1*. Only group nodes are shown in this figure. 66

Structural model for fuselage of *vehicle1*. 67

Modeling approach for lesson 2. 68

Tree for *vehicle2*. Compressed nodes shown in reverse video. 69

Expanded “wing” node. 70

Illustration of screen after step 6 of lesson 2. 70

Stations are only at spar locations. 70

Step 8 of lesson 2 adds wing-connection breaks to the tip-fitting. 71

Structural model for tip-fitting of *vehicle2*. 72

Illustration of screen after step 15 of lesson 2. 72

Illustration of screen after step 16 of lesson 2. 73

Illustration of screen after step 17 of lesson 2. 73

Illustration of screen after step 18 of lesson 2. 74

Structural model for fuselage of *vehicle2*. 75

The *merge fuselage* Option is used to correct problem area where wing intersects the fuselage. 75

PREFACE

About the Software

Version 3.0 of the Solid Modeling Aerospace Research Tool (SMART) software package adds three new options to the main menu, S-WING, S-FUSE and MERGE. These options, referred to collectively as SMART Structures, were added in order to address the geometric modeling needs of the aerospace structural engineer.

SMART Structures simplifies construction of both external and internal geometries that are suitable for finite-element model mesh generation. It provides a highly interactive and dynamic capability for generating structural geometries with Bezier bicubic patches. Features include the generation of structural elements for wings and fuselages, the integration of wing and fuselage structural assemblies and the integration of fuselage and tail structural assemblies.

SMART was developed by the Vehicle Analysis Branch at NASA Langley Research Center (LaRC). It is also used by a number of organizations at LaRC, other NASA Centers, and at some aerospace corporations. Because the code contains calls to Graphics LibraryTM(GL) routines, SMART must be run on a Silicon GraphicsTM workstation. The workstation should have at least a 19 inch screen. For information about obtaining the software, you can send electronic mail to smart@vab01.larc.nasa.gov or send requests to the following address: Mail Stop 365, NASA Langley Research Center, Hampton, VA 23681-0001.

About the Guide

This SMART Structures User's Guide includes primarily information that is new for SMART version 3.0. Previous experience with SMART is assumed. You should know:

- the layout of the screen and use of menus,
- the various options available for viewing the created geometry,
- how the tree graph is used to visually represent the relationship between the many objects in a model,

- how to modify the appearance of your model, make certain parts of the drawing visible or invisible, and adjust the coloring of your model and the shading of its parts,
- how to edit individual patches within a surface,
- how to save objects in data files and how to retrieve them for later editing.

In addition to these features, which are discussed in the SMART User's Guide Version 2.0, it is also assumed that you have an understanding of aeronautical and aerospace design principles.

This guide begins with a description of the basic concepts and capabilities of the software and then covers the menus and other user-interface options that are available for constructing a structural model. This guide is divided into the following sections:

- Chapter 1 describes the concepts and capabilities of SMART Structures.
- Chapter 2 describes the menus and other user-interface options that are available for constructing a structural wing (or vertical tail or horizontal tail or tip fin, etc.)
- Chapter 3 describes the menus and other user-interface options that are available for constructing a structural fuselage.
- Chapter 4 describes how to correct a surface so that all patches share vertices along common edges.
- Appendix A describes how to bring in a wing that was created on a system other than SMART.
- Appendix B describes constraints on geometric input to the structural fuselage.
- Appendix C is a tutorial.

Convention

You use a three-button mouse to run SMART.

When you read:	You should:
Click or left-click	Click the left mouse button.
Middle-click	Click the middle mouse button.
Right-click	Click the right mouse button.

CHAPTER 1

Overview

Who is SMART Structures for?

SMART Structures is for the structural engineer who wants to apply finite element analysis (FEA) methods to conceptual and preliminary-level aerospace designs. It is a software tool that lets the engineer move quickly from a geometry that defines a vehicle's external shape to one that has both internal and external components. (See fig. 1.1.) The output is a geometry that is suitable for finite-element model mesh generation.

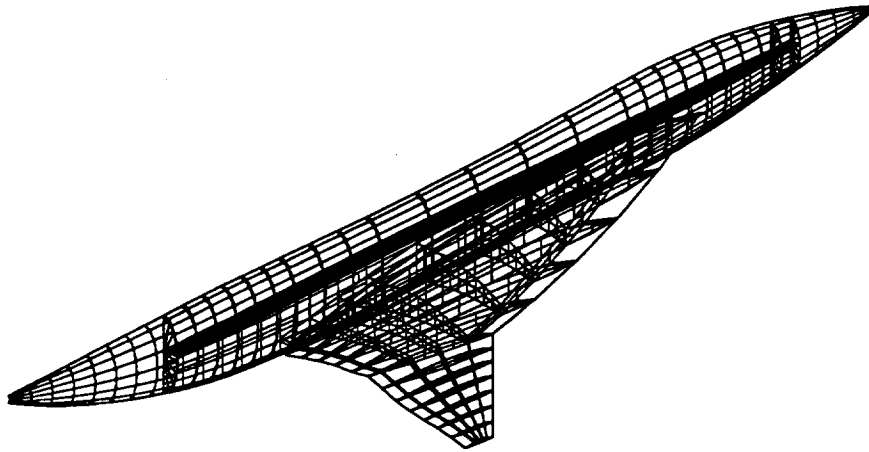


FIGURE 1.1.

Illustration of model generated by SMART Structures.

What kind of capabilities does SMART Structures provide?

What makes SMART Structures unique is that it offers modeling capabilities that are specific to aerospace vehicles. Using a point and click interface, the user specifies where to put ribs and spars in the wing; divides the fuselage into topological subcomponents such as fore, mid and aft assemblies; and places variable depth ring frames, longerons, keel beams, and floor structure within each assembly. At any stage in the process, a geometry for the vehicle that is consistent with FEA requirements and that includes integrated wing and empennage carry-through and frame attachments can be generated. Structural layouts are saved in files, so that the structural components can be recalled, edited and output again in edited form. The resulting reduction in the interactive modeling time means that greater use of FEA can be made at an earlier stage in the design process. The ability to edit the structural lay-

out can make what were previously major design modifications, such as relocating the wing relative to the fuselage or changing the number of ring frames, a much easier task.

How does SMART Structures interface with other programs?

Importing the vehicle's external shape into SMART Structures

There are two ways to introduce the external shape of a vehicle into SMART Structures. One is to use other modules of SMART to generate the external shape. The other is to read in the shape geometry (as a file) from an outside CAD system. Among the file formats that SMART is capable of reading, the PATRAN TMneutral file is the only format that is useful for importing a vehicle's external shape. See chapter 12 of the SMART User's Guide, Version 2.0, for more information about SMART's capability for reading and writing PATRAN files.

Exporting the SMART Structures Model for finite-element mesh generation

SMART Structures creates surface models of structural components. Because these surface models satisfy FEA requirements, they can be transferred to a finite element program and be easily meshed.

PATRAN neutral format

The PATRAN neutral file is the format that is provided for exporting the SMART Structures model to a finite element program. The SMART model is transferred to PATRAN as a phase-I geometry. See chapter 12 of the SMART User's Guide, Version 2.0, for more information.

.map file

Although it is not mentioned in the SMART User's Guide, Version 2.0, a second file is automatically created when SMART writes a PATRAN neutral file. This file lists the id numbers of the phase-I packet types (cubic curves and bicubic patches) that are output for each component. The file's name is ".map" appended to the name of the PATRAN file.

How Does SMART Structures interface with other SMART modules?

Figure 1.2 shows the interfaces between SMART Structures and other SMART modules. The basic interface to the rest of the SMART is through the data tree. The tree, which is presented graphically on the screen, is a mechanism for storing component geometry and for grouping components together

to form more complex assemblies. All geometry is stored in the tree as either bicubic curves or bicubic patches. The user indicates which geometry is to be edited by selecting nodes from the tree. When the user enters SMART Structures, he or she selects a surface that defines the external shape of the wing or fuselage from the tree. This geometry will have been created by defining an aero-specific (wing, tank, etc.) primitive in PRIMITIVES Mode or by importing a wing or fuselage into SMART by reading its geometry from a data file. When the user exits SMART Structures, the resulting structural model is inserted into the data tree and is thus available to other SMART modules such as visualization. To output the structural model, the user enters FILE Mode and outputs the model as phase-I geometry to a PATRAN neutral file format.

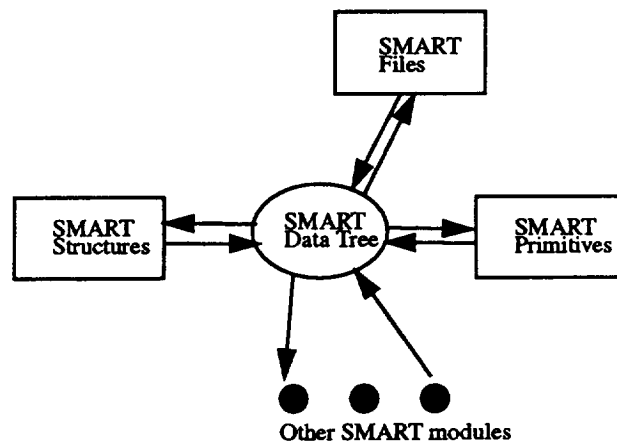


FIGURE 1.2. Data flow relative to SMART Structures.

Using SMART to generate a structural model

Wing

The purpose of the S(tructural)-WING editor is to develop structure for wings, vertical tails, horizontal tails and tip-fins. Figure 1.3 diagrams the modeling approach of the S(tructural)-WING editor. The

same approach is used for all types of wing. Hereafter, this manual uses “wing” to refer to all types of wing-like objects.

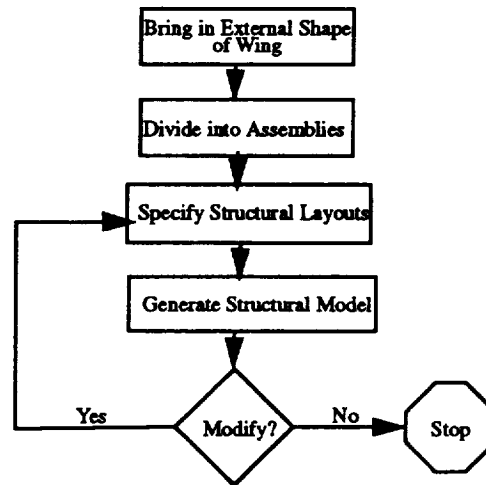


FIGURE 1.3. Process for modeling the structural wing.

Step 1 brings in the external shape of the wing. Step 2 divides the wing into one or more assemblies. Step 3 specifies the structural layout for each assembly; that is, its wing box, and rib and spar patterns. Step 4 uses the structural layouts defined in the previous step to generate the structural model. While not shown in figure 1.3, the user can save the layout of the structural wing in a file at any time during the modeling process. A saved layout consists of the planform definition, the rib and spar specifications, and any other information that is needed to re-create the structural model.

The basic input is Bezier bicubic patches defining the shape of the upper and lower wing. The program automatically determines a planform from the patches that define the upper wing. Streamwise chord-lines are inserted into the planform wherever slope discontinuity is detected along the leading or trailing edges. As shown in figure 1.4, the chord-lines divide the wing into assemblies. A planform editor enables the user to change the default assembly divisions by adding or subtracting chord-lines.

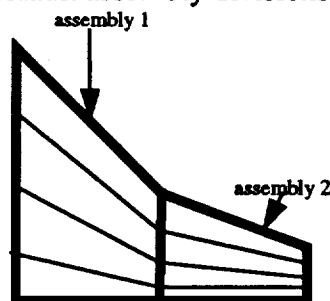


FIGURE 1.4. Streamwise chord-lines mark assembly divisions.

Each assembly defines a wing box made from the leading edge spar, the trailing edge spar, the root rib and the tip rib. The leading-edge spar is the major load-bearing structure located behind the wing's leading edge, and the trailing-edge spar is the major load-bearing structure located in front of the wing's trailing edge. The root rib is the inner-most rib, and the tip rib is the outer-most rib. The user is provided with a variety of options for defining and editing 1) the leading- and trailing-edge spars; 2) rib structure for the region immediately preceding the leading edge spar and following the trailing edge spar; and 3) rib and spar structure for the region inside the wing box.

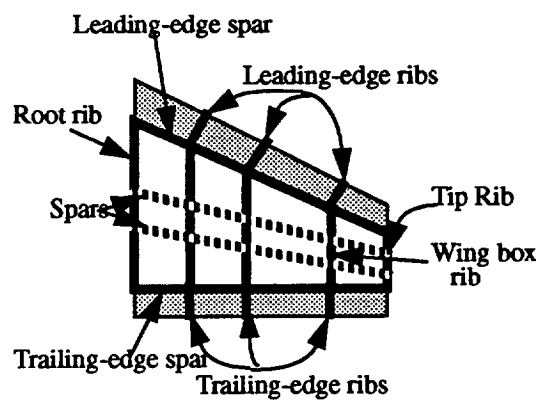


FIGURE 1.5. Wing planform and internal structure.

Figure 1.5 illustrates typical positions for the leading edge spar, the leading edge ribs, the trailing edge spar, the trailing edge ribs, wing box ribs, and wing box spars. Figures 1.6 shows some of the options that are provided for defining spars and rib structure. If more than one assembly is defined, the software provides for automatic matching of leading- and trailing-edge spars at the common chord.

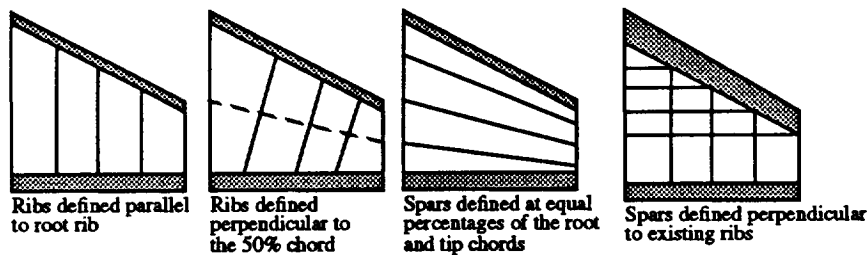


FIGURE 1.6. Some options for defining rib and spar structure.

When the structural model is generated, two-dimensional geometry elements are created for both the wing surface and the user-defined ribs and spars. These elements are represented by bicubic Bezier patches. The new surface is reformulated from the original geometry so that the edges of its patches

correspond to the underlying rib and spar elements. The resulting structural model is added back to the SMART data tree for use by other SMART modules.

Fuselage

Figure 1.7 diagrams the modeling approach of the S(tructural)-FUSE(lage) editor.

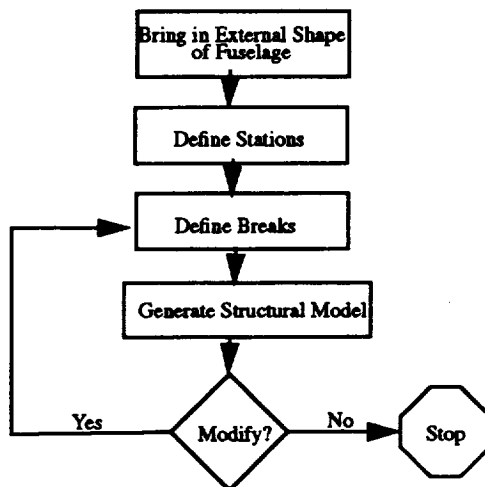


FIGURE 1.7. Modeling process for structural fuselage.

Step 1 brings in the external shape of the fuselage. Step 2 specifies stations along the fuselage's longitudinal axis. Stations identify where ring frames are located, where bulkheads are located, and where assembly divisions occur. Step 3 adds lateral cuts along the longitudinal axis which define longerons, flooring and keel beams. Hereafter, this manual refers to lateral cuts as breaks. Step 4 uses the stations and breaks that were defined to create the structural model. The process is iterative, allowing the user to modify stations and breaks until he or she is satisfied with the structural model. The user can save the structural layout of the fuselage at any step in the process.



FIGURE 1.8. Stations are placed perpendicular to the axis of the fuselage.

As shown in figure 1.8, stations are placed perpendicular to the longitudinal axis of the fuselage. The user enters station locations by either pointing at a side view of the fuselage, by numeric input, or by designating that stations should occur at a uniform intervals (for example, every 20 inches). Not all

stations are user-defined. For instance, the software automatically adds stations where the spars of the structural wing intersect the fuselage.

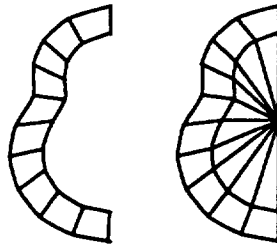


FIGURE 1.9. Examples of ring frame and bulkhead structure.

Ring frames can be generated as one- or two-dimensional elements. Two-dimensional ring frame elements, as illustrated in figure 1.9, are referred to as deep frames. Depth may vary linearly from the top to the bottom of the ring frame. Also in figure 1.9 is an example of a bulkhead. The user defines the boundaries of an assembly by designating certain stations to be assembly dividers. Assemblies allow the placement of structural elements in the fuselage to differ depending on the position along the longitudinal axis of the fuselage. For example, a typical breakdown of the fuselage is into forward, mid and aft assemblies.

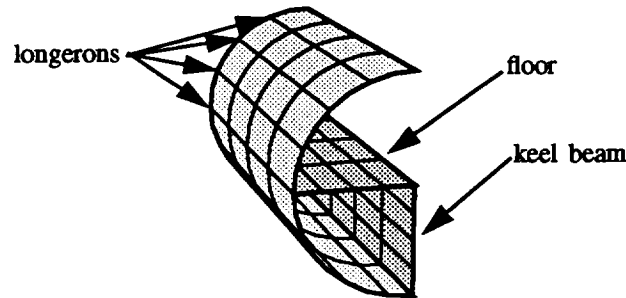


FIGURE 1.10. Typical placement of longeron and beams.

Longerons, flooring and keel beams are added through a cutting plane user interface. Figure 1.10 illustrates typical placements of longerons and beams. User input is captured and stored until the fuselage is defined. Upon finishing, two-dimensional geometry elements are created for the fuselage surface, the ring frame, bulkhead and beam elements. One-dimensional elements are created for the longeron elements. As with the wing, elements in the fuselage surface are reformulated from the input surface model, so that output patches are defined by the boundaries of the underlying structural elements. Two-dimensional elements are represented by bicubic patches and one-dimensional elements are represented by cubic curves. The fuselage's structural model is added back to the SMART data tree for use by other SMART modules.

Wing-Fuselage Integration

Figure 1.11 illustrates different methods for joining wing-spar structure to fuselage-ring frame structure. All of these methods are supported by SMART Structures. The wing can be oriented so that its upper surface is completely above the fuselage (high-wing), its lower surface is completely below the fuselage (low-wing), or both surfaces intersect the fuselage (mid-wing). The two cases of the mid-wing illustrate the difference between *carry-through* structure and *wing-connection* structure. Carry-through is continuous through the fuselage causing the wing box area to be cut out from the fuselage's skin. In contrast, wing-connection spars attach directly to the fuselage frames. Both options, carry-through and wing-connection, can also be applied to high- and low-wings.

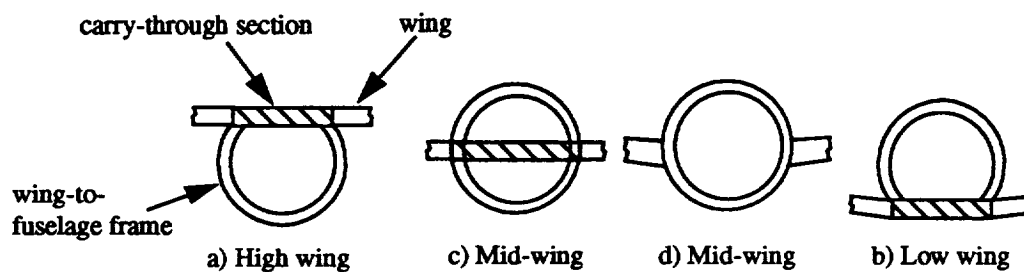


FIGURE 1.11. Methods for joining spar and ring frame structure.

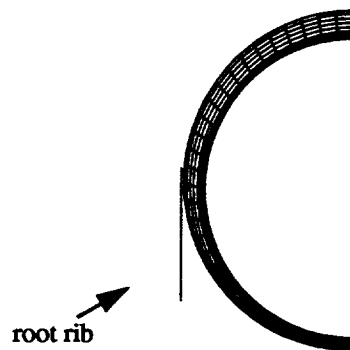


FIGURE 1.12. Side-view of wing root rib and fuselage body.

Placement of the wing relative to the fuselage is provided jointly by the structural wing editor and SMART's PACKAGING Mode. One constraint, as shown in figure 1.12, is that the root rib of the wing must be completely outside the fuselage. In SMART's structural wing editor, you can use the planform editor to move the root rib outwards (see fig. 1.13.) If necessary, the structural wing can be

rotated, translated or scaled in PACKAGING Mode. As show in figure 1.14, the vertical tail is an example of a rotated wing.

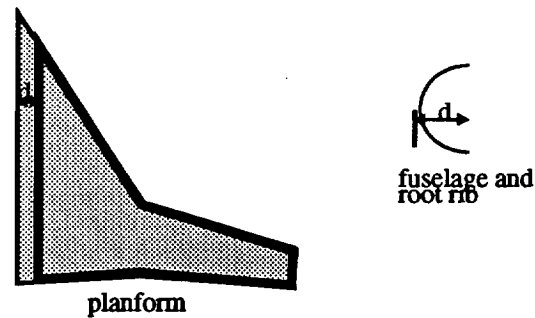


FIGURE 1.13. Inside the planform editor, the root rib is moved outwards from the center line.

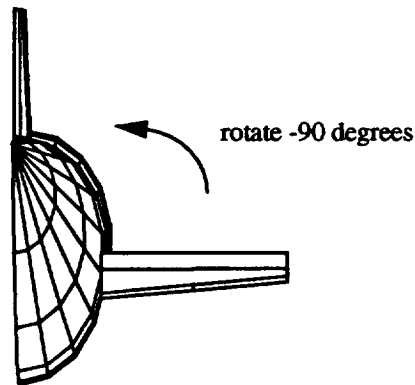


FIGURE 1.14. The vertical tail is an example of a rotated wing.

Tank-Fuselage Integration

Upon entering the S-FUSE editor, the user can bring in tanks that are internal to the primary fuselage geometry. This allows the user to define deep ring frames that project inward from the fuselage's outer surface until they meet the tank surface. To generate a structural model for the tank, SMART

reformulates the patches of the initial tank surface so that the ringframes and longerons of the outer fuselage are projected radially to the tank surface. (See fig. 1.15.)

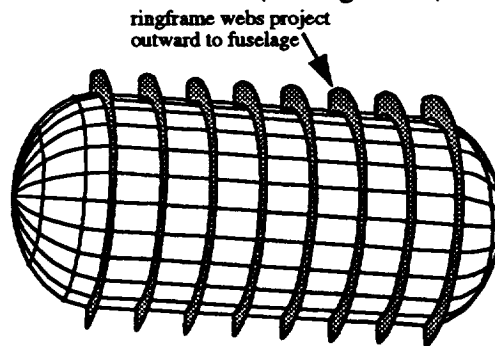


FIGURE 1.15. Illustration of tank generated by SMART Structures

Mesh Alignment

To be consistent with FEA requirements, it is necessary that patches share vertices along common edges. The surfaces generated by S-FUSE and S-WING Modes do not always meet this constraint. Unmatched vertices typically occur at the juncture of two fuselage or wing assemblies, and where the structural wing joins the fuselage's carry-through structure. The MERGE resolves these problem areas by splitting existing patches or inserting transition patches.

CHAPTER 2**S-WING**

This chapter describes how to develop a wing structure that can be used for finite-element mesh generation.

The Process

You follow a four-step process to build the structural model: (1) define the wing box, (2) create a rib pattern, (3) create a spar pattern, and (4) generate the wing surface. The process is iterative, allowing you to modify rib and spar patterns until you are satisfied with the structural model.

Edit Mode	Purpose
wing box	position leading and trailing edge spar
ribs	create, modify or delete ribs
spars	create, modify or delete spars
surface	generate patches for the structural model
library	save layout of ribs and spars in a file

TABLE 2.1. Edit mode of S-WING.

Table 2.1 lists the edit modes of S-WING. The first four edit modes correspond to the four steps of the modeling process. The last edit mode, library, can be entered at any time during the modeling process. It is for saving the layout of the structural wing in a file.

Input Geometry

To access the options for creating a structural wing geometry, select S-WING from the main menu. SMART prompts you to select a wing from the Tree. (See Appendix A if the selected wing is NOT a group node generated by the PRIMITIVE mode of SMART 3.0.) After the wing is selected, its planform appears in the right View Window. If it is a multi-sectioned wing, there will be a planform for each wing section.

Display of the Wing Planform

Figure 2.1 shows the planform of one wing section. The planform is the outline of the X_Y projected area of the wing section. The dashed lines that run span-wise across the planform are called spars, and the solid lines that run chord-wise across the planform are called ribs. The wing box, which is shown

by the thick lines, is defined by the leading-edge spar, the trailing-edge spar, the root rib and the tip rib.

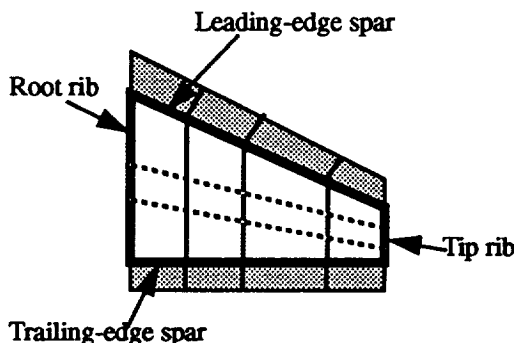


FIGURE 2.1. Planform and wing box of one wing section.

Active Wing Section

Although the planforms of all wing sections are displayed on the screen simultaneously, only one wing section can be modified at a time. A slider bar is provided for selecting the wing section that you want to edit. SMART highlights the active section by outlining it in white; inactive wing sections appear grayed-out.

Wing Box

This section describes how to position the leading edge and trailing edge spar. To access these options, select *wing box* from the Edit Mode menu. By default, the leading-edge spar is coincident with the planform leading edge and the trailing-edge spar is coincident with the planform trailing edge. The root and tip rib cannot be moved. Figure 2.2 shows the menus for editing the wing box.

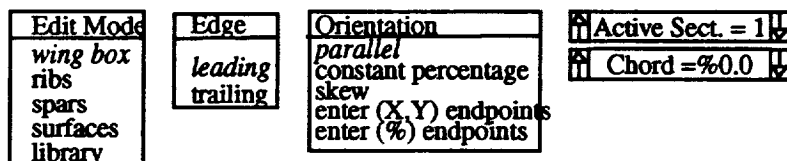


FIGURE 2.2. Menus for defining the wing box.

Selecting Edge-spar to Edit

Together, the Active Section slider bar and the Edge menu determine which edge you are editing (see fig. 2.3). The slider bar specifies which wing section. The menu specifies whether you are moving the leading or trailing edge spar of that section.

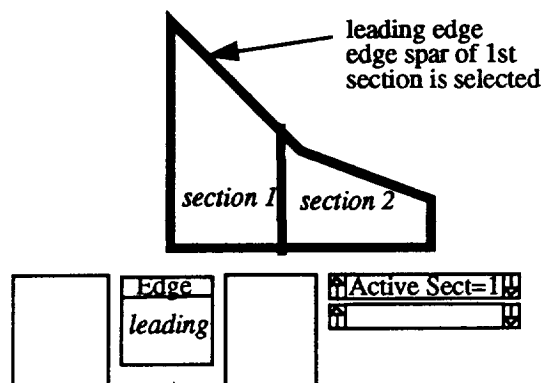


FIGURE 2.3. The Edge menu and the Active Section slider bar specify which edge to edit.

parallel

To move an edge-spar so that it remains parallel to the planform edge, select *parallel* from the Orientation menu and adjust the value on the Chord slider bar. This value, which is a percentage of the root chord length (the leading edge is 0 percent and the trailing edge is 100 percent), specifies how far back or how far forward of the planform to place the edge-spar. An alternative to using the slider bar is to left-click on the root chord where you want the edge-spar located.

constant percentage

The *constant percentage* option constrains the edge-spar so that it intersects the root and tip chords at equal percentages of their chord lengths. *Constant percentage* edge-spars are entered the same way as *parallel* edge-spars.

skew

The *skew* option provides two slider bars for placing the edge-spar. One bar specifies a percentage length of the root chord and the other bar specifies a percentage of the tip chord. Alternatively, the mouse can be used to place the edge-spar: point the mouse cursor over the root chord where you want the edge-spar located, press the left mouse button, move the cursor to the edge-spar's tip chord location and release.

enter (xy)endpoints/ enter (%) endpoints

The *enter (xy) endpoints* option lets you specify the edge-spar by entering the X- and Y-coordinates of its two endpoints. The other keyboard option, *enter(%) endpoints*, is similar except that you specify endpoints as a percentage of the root chord length and a percentage of the wing box's span. Do not put commas or other punctuation between the entered values.

When You Have More Than One Wing Section

With multi-sectioned wings it is best to use the slider bars to position the leading-edge and trailing-edge. When you use slider bars, the software provides for automatic matching of the leading- and trailing-edge spars at the common chord. When the spar is placed using the mouse or the keyboard there is no automatic updating of the adjacent sections.

Caveat

If you try to change the wing box after ribs or spars have been entered, SMART asks you whether you want to delete the existing ribs and spars. If you answer no, you cannot edit the wing box.

Ribs

This section describes how to create, move and delete ribs. To access these options, select *ribs* from the Edit Mode menu. Figure 2.4 shows the menus for specifying ribs. Each wing section is divided into 3 regions: the leading edge box (precedes the leading edge spar), the wing body (inside the wing box), and the trailing edge box (follows the trailing edge spar). Ribs can be entered into all of the three regions. As shown in figure 2.5, you use the Active Section slider bar to select which wing section to edit and the Box menu to specify which region within that section.

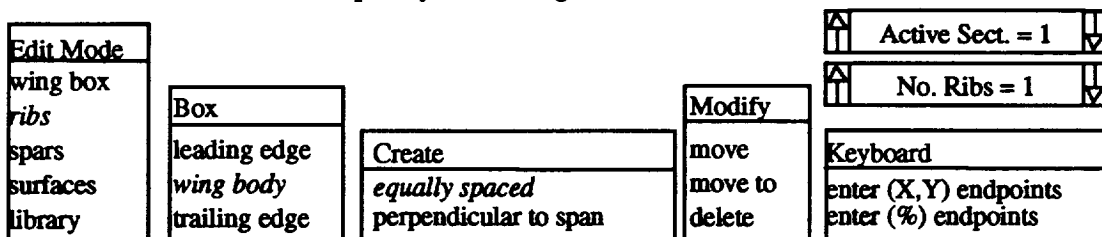
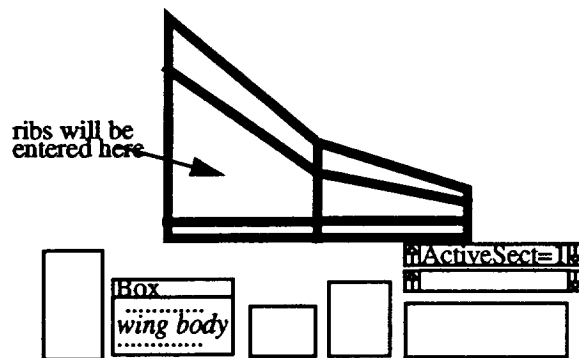


FIGURE 2.4. Menus for ribs.

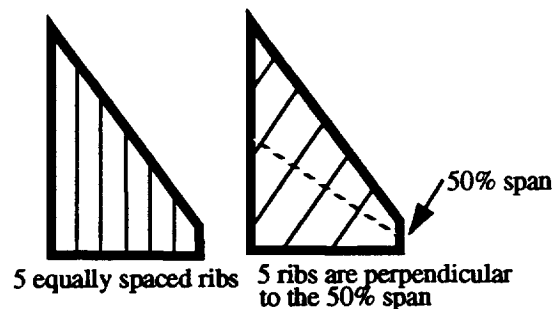
**FIGURE 2.5.**

Together, the Box menu and the Active Section slider bar determine which part of the wing you are editing.

equally spaced / perpendicular to chord

To create ribs, you first select a rib orientation from the Create menu. The two orientations, *Equally spaced* and *Perpendicular to span*, are illustrated in figure 2.6. *Equally spaced* creates streamwise ribs. *Perpendicular to span* creates ribs that are perpendicular to a percentage span. Percentage span refers to a line that intersects the root and tip chords at a specified percentage of their respective lengths. A slider bar, *Span%*, is provided for specifying the percentage. For both types of ribs, you can use the *No. Ribs* bar to enter a number of equally spaced ribs or you can use the mouse to enter ribs one at a time. To create a rib with the mouse, point the mouse cursor over the leading or trailing edge spar and left-click.

Note: Adjusting the *No. Ribs* slider bar after adding ribs using the mouse or keyboard causes the ribs to revert to equal spacing.

**FIGURE 2.6.**

Examples of *Equally spaced* and *Perpendicular to span*.

enter (X,Y) endpoints / enter (%) endpoints

The two keyboard options, *enter (xy) endpoints* and *enter(%) endpoints*, work exactly the same as when defining the wing box.

move

To move a rib endpoint, select *move* from the Modify menu and move the mouse cursor into the right View Window. The rib endpoints are marked with small squares. Position the mouse cursor on the endpoint that you want to move. Press the left mouse button, drag the endpoint to a new location on the wing box, and release. The action is ignored when it causes ribs to cross or when you move the endpoint off the wing box.

Note: If the square marking an endpoint is yellow, then the endpoint is shared with another rib or spar.

move to

The *move to* option sets the endpoint of one rib equal to the endpoint of another rib. Move the mouse cursor into the right View Window, and click on two endpoints. The first endpoint is set equal to the second endpoint.

delete

To delete a rib, select *delete* from the Modify menu and click on either of the rib's endpoints. To delete all the ribs, click on the *Delete All* button.

Spars

Figure 2.7 shows the menus for specifying spars. The menus are the same as for ribs except that different options are presented on the Create menu. Figure 2.8 shows examples of the spar options.

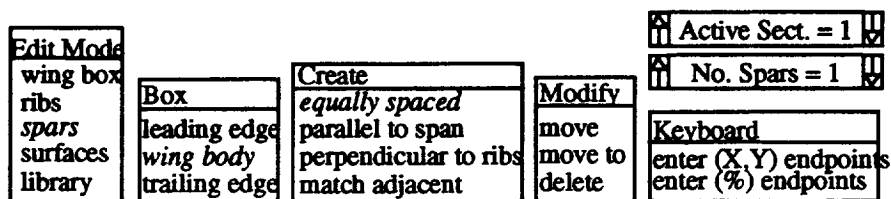


FIGURE 2.7. Menus for spars.

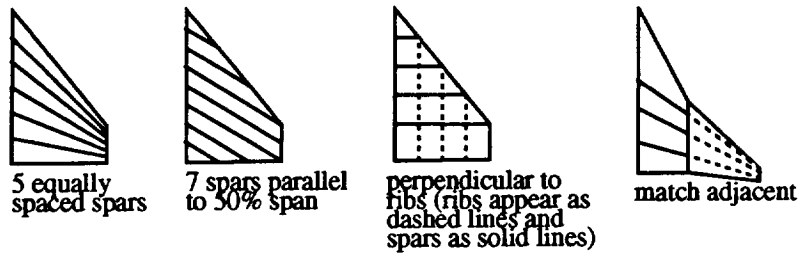


FIGURE 2.8. Examples of spar options.

equally spaced/parallel to span

The *equally spaced* option creates spars that are equally spaced down the length of the root and tip ribs. The *parallel to span* option creates spars that are parallel to a percentage span. The *span%* bar specifies the percentage. For both options, you can use the *No. Spars* bar to enter a number of equally spaced spars or you can use the mouse to enter spars one at a time.

perpendicular to ribs

The *perpendicular to ribs* option creates spars that are perpendicular to existing ribs at the points where the ribs intersect the leading-edge spar. To create ribs one at a time, click on the endpoint of any rib. To create a spar for each of the ribs, click the ALL button which is at the top of the View Window.

match adjacent

The *match adjacent* spar creates spars in the active section that are slope continuous with the spars in adjacent wing sections. SMART fails to add a spar if it would intersect one or more of the existing spars.

Surfaces

To access the options for generating the structural surface, select *surfaces* from the Edit Mode menu. Figure 2.9 shows the menu for generating the structural surface. Selection of *wing section* generates

the surface for only the active wing section; *entire wing* generates the surfaces for all of the wing sections.

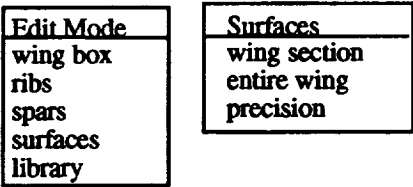


FIGURE 2.9. Menus for generating the structural surface.

SMART uses the definition of the wing box, rib pattern and spar pattern to calculate a wing section. First, each skin element is checked. Skin elements are the quadrilateral or triangular surfaces which are created by the intersections of ribs and spars. For instance, the wing shown in figure 2.1 contains 20 elements (12 elements in the wing body, 4 elements in the leading edge You will get an error message if any of the elements have more than four sides. Five- and six-sided elements are highlighted in red. Highlighted elements, which appear to be four-sided, actually have five sides, but one side is too small to be seen on the screen. If you get an error message, you must edit the rib or spar pattern so that you have no five- or six-sided elements.

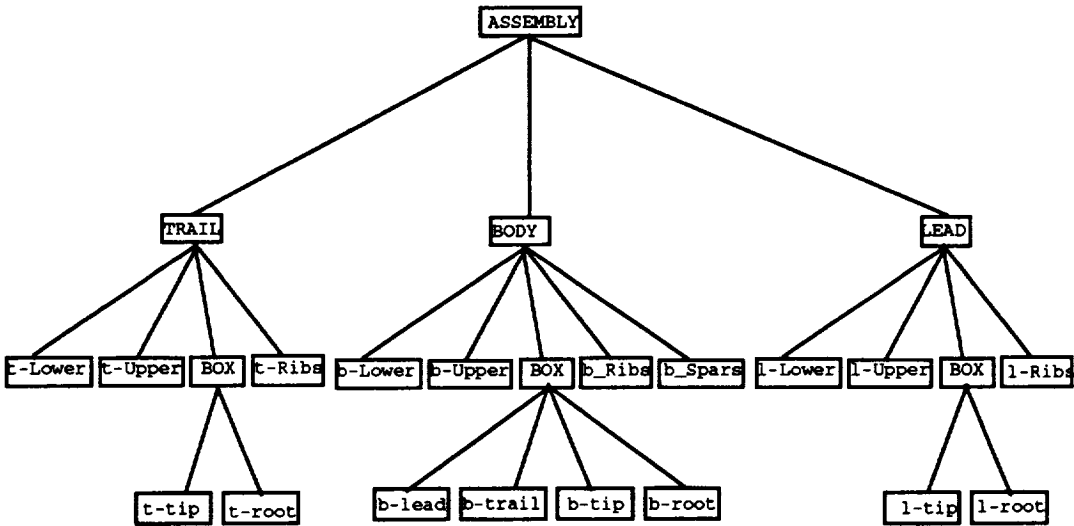


FIGURE 2.10. Tree that is created for each assembly.

If no element has more than four sides, SMART prompts you to enter a name and a two-letter identifier for your wing. The wing is resurfaced so that the upper and lower patch boundaries coincide with

the underlying internal structural rib and spar elements. As shown in figure 2.10, calculation of a wing section creates a hierarchy of components. At the top level, the wing section is divided into three groups: the leading edge region, the wing body, and the trailing edge region. Each of the three groups further break down into an upper surface, a lower surface, rib components, spar components and a wing box. The wing box of the wing body is composed of a leading edge spar, a trailing edge spar, a root rib and a tip rib. The wing box of the leading and trailing edge regions is composed of just the root and tip ribs.

By default, SMART generates one patch per skin element. For instance, a total of 40 surface patches would be generated from the wing planform shown in figure 2.1. The upper and lower surfaces for the wing body would contain 24 surface patches and 8 surface patches would be created for both the trailing and leading edge sections. The *precision* option lets you change the number of patches that are generated for each element in a wing section. SMART prompts you to enter two numbers, the number of patches between ribs and the number of patches between spars. If you entered 2 between spars and 3 between ribs, then SMART would generate 144 patches for the wing body section shown in figure 1.5, six patches per element. Forty-eight patches would be created for both the trailing and leading edge sections.

The same surface generation algorithm is used for all types of wing-like objects. Consequently, an extraneous surface is generated in the X-Z plane when the wing is a vertical tail. (See fig. 2.11.) To correct this problem, you can enter SMART's Tree mode and delete the nodes which contain the lower surface. For example, you would delete "t-Lower", "b-Lower", and "l-Lower" from the tree illustrated in figure 2.10.

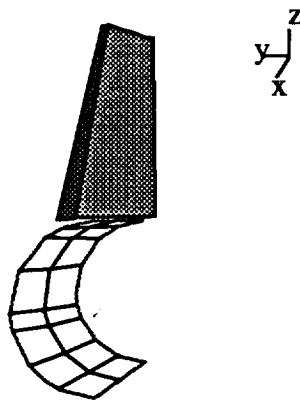


FIGURE 3.11. Unnecessary surface is generated in X-Z plane for vertical tails.

Library

You can save your work in a file. These files have a “.fem” extension and are stored in your working SMART directory. Operations for reading, writing and deleting “.fem” files are available when you select *library* from the Edit Mode menu. As shown in figure 2.12, you are provided with a menu of file operations.

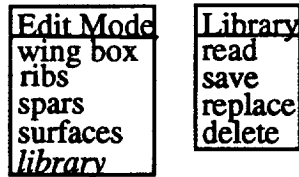


FIGURE 2.12. Menus for writing or reading the structural wing to a file.

read

To read a file, select *read* from the File Options menu and select a file to read from the File Chooser. SMART checks that the planform of the saved wing definition matches the planform of the current wing. If the two planforms don't match you will get an error message. Otherwise, any existing ribs and spars are overwritten by the ribs and spars which are read from the file.

save

To write a file, select *save* from the File Options menu. At the prompts, type in a file name and a brief file description (80 characters or less). SMART writes the current wing box definition, rib pattern and spar pattern to the file.

replace

The *replace* option is similar to the save option except that you chose an existing file.

delete

The *delete* option lets you remove files.

CHAPTER 3

S-FUSE

This chapter describes how to develop a fuselage geometry that can be used for finite-element mesh generation.

Bringing the External Shape into S-FUSE

To bring in the external shape of the fuselage, select S-FUSE from the main menu and then indicate the fuselage geometry by picking a node from the Tree. After the fuselage is selected, additional selections from the Tree bring in root ribs, domes and barrels. Root ribs allow you to create structure that integrates wing and fuselage structural components. Domes and barrels allow you to add internal tanks to the structural model. When you are done making selections from the Tree, move the mouse cursor into the Menu Display Area.

The fuselage surface and component geometries that you input to S-FUSE need to satisfy certain constraints. For example, root ribs must be completely external to the fuselage. In addition, SMART assumes information about how a wing's root rib is generated in S-WING mode in order to determine spar locations. If you pick a root rib that was not generated in S-WING mode, then it must satisfy the geometric constraints listed in Appendix B or you will get an error. Appendix B also describes restrictions on the fuselage and tank geometries.

Root ribs allow you to create structure that integrates wing and fuselage structural components. When a root rib is input, SMART creates an upper and a lower wing surface by extruding the root rib to the fuselage's center line. To confirm that SMART has extruded the root rib the correct distance and direction, you are prompted to validate the surfaces that appear on the screen by selecting *yes* from the Verify menu. If the extruded surface does not intersect the fuselage, select *no* and enter a correct distance. Hint: to switch the direction that the root rib is extruded, enter the negative of the default extrusion value.

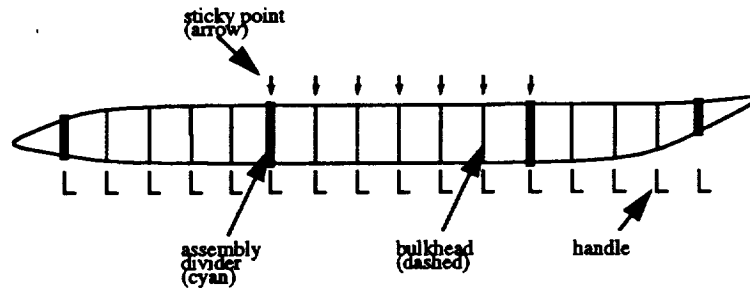


FIGURE 3.1. Fuselage display.

Display of the Fuselage

As shown in figure 3.1, the fuselage is displayed in the right View Window as an outline of its X-Z projected area. The lines perpendicular to the longitudinal axis of the fuselage are called stations. Cyan-colored stations partition the fuselage into assemblies and are called assembly-dividers. Bulkheads are represented on the screen as dashed lines. Stations that are neither an assembly-divider nor a bulkhead appear as solid yellow lines.

The L-shaped line below each station is its handle. When the mouse cursor is on a handle, the word Station and the station's X-coordinate are printed in the screen area above the right View Window. If the mouse cursor is on the fuselage axis but not over a handle, the cursor's X-position on the fuselage axis is printed but not the word Station. A station is selected by pointing the mouse cursor over the station's handle and clicking.

Sticky Points

The arrows in figure 3.1 denote sticky points on the fuselage's axis. Stations fall into 2 categories: 1) those that are created by the user, and 2) those that are derived from the input geometry. If a station is "derived", its original position is recorded as a sticky point on the fuselage axis. If a derived station is accidentally moved, sticky points make it easy to later move the station back to its original position. When the mouse cursor is close to a sticky point, the cursor automatically resets to the exact value of the sticky point. Selection of a dome cap or a tank barrel causes stations and sticky points to be added at the minimum and maximum X-values of the dome or barrel. Selection of a wing's root rib adds stations and sticky points where the wing's spars intersect the fuselage.

As shown in table 3.1, the color of the arrow indicates why the sticky point was added.

color	indicates
green	spar-fuselage intersection
magenta	tank boundary
blue	dome cap
black	wing-tank intersection
red	wing surface drops below or above fuselage

TABLE 3.1. Color of arrow indicates origin of sticky point.

The Process

Once the input geometry is read in, you follow a three step process to build the structural model: (1) define stations, (2) define breaks, and (3) generate the model. The process is iterative, allowing you to modify stations and breaks until you are satisfied with the structural model. .

Edit Mode	Purpose	examples
stations	define structural members that run from top to bottom	ringframes, bulk-heads
breaks	define structural members that run from front to rear	longerons, flooring, keel beams, carry-through
surface	generate patches for the structural model	
library	save layout of stations and breaks in a file	

TABLE 3.2 Edit modes of S-FUSE.

Table 3.2 lists the edit modes of S-FUSE. The first three edit modes correspond to the three steps of the modeling process. The last edit mode, library, can be entered at any time during the modeling process. It is for saving the layout of the structural fuselage in a file.

Stations

This section describes how to add, move and delete stations, how to distribute stations between two existing stations, how to assign offset-depths for creating ring frame webs, how to parti-

tion the fuselage into assemblies, and how to specify bulkheads. To access these options, select *stations* from the Edit Mode menu. Figure 3.2 shows the menus for defining stations.

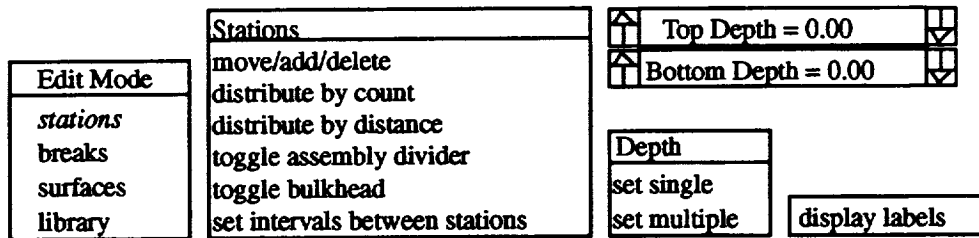


FIGURE 3.2. Menus for defining stations.

HINT: As you execute the various options, you may find that stations become bunched so close together that they cannot easily be selected with the mouse. If this happens, use the X-ZM View Option to zoom and pan the fuselage or 1-VU to increase the size of the right View Window.

move/add/delete

When *move/add/delete* is selected from the Stations menu, the left mouse button moves stations, the middle mouse button adds stations and the right mouse button deletes stations. (See table 3.3.) Holding down the Shift Key as you depress the left or middle mouse button allows you to type in an X-coordinate.

Action	Mouse Button	Usage
move	left	Point mouse cursor over station's handle, press button, drag the station to its new position and release.
	with Shift-key	Instead of dragging the station, you are prompted to enter an X-coordinate.
add	middle	Move the mouse cursor to where on the fuselage's axis you want the station and click mouse button.
	with Shift-key	Move cursor to approximate position on fuselage axis, click mouse button and enter X-coordinate of new station.
delete	right	Move cursor over station handle and click mouse button.

TABLE 3.3 The *move/add/delete* option.

TABLE 3.4**distribute by count/by distance**

For evenly spaced stations, pick either *distribute by count* or *distribute by distance* from the Stations menu. Select with the left mouse button the two stations to distribute between. Depending on which option you picked, enter either the number of stations to distribute between the two selected stations or a unit distance (a station is added every x.x units).

toggle assembly divider

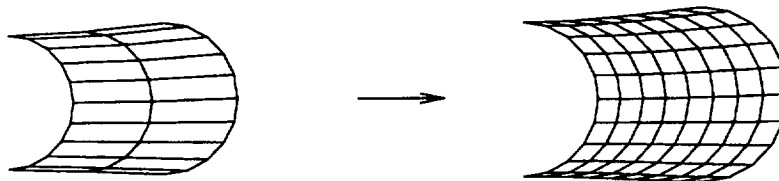
Use this option to partition the fuselage into assemblies. Selecting a station toggles its divider status on (cyan) or off (yellow).

toggle bulkhead

Use this option to specify bulkheads. Selecting a station toggles its bulkhead status on (dashed line) or off (solid line).

set intervals-between-stations

This option increases the number of patches that are created when you generate the structural fuselage. Figure 3.3 shows wireframes of two structural models that were generated for the same assembly. By default, SMART generates models like the one on the left. The span between each two stations is represented by one column of patches. Intervals-between-stations allows you to increase the number of patch columns. The right-hand wireframe was generated with intervals-between-stations set to four. To define intervals-between-stations for an assembly, select this option from the Stations menu and click on the handle of the assembly's divider station. At the prompt, enter the desired number of columns.

**FIGURE 3.3.**

Both models have the same number of frames (3) but different intervals-between-stations specifications.

set single

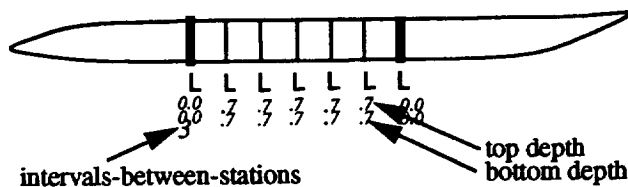
Depths are assigned to stations in order to generate ring frame webs. To change a station's depth, select single from the Depth menu. Use the slider bars to specify desired depths. There are two slider bars because depth may vary linearly from the top to the bottom of the station. Selecting a station assigns the indicated depths.

set multiple

To assign a depth to two or more stations at a time, select multiple from the Depth menu. Then select a start and a stop station. Depths are assigned to the selected stations and the stations that are between them.

display labels

As shown in figure 3.4, station depths are printed in the right View Window when this option is on. The intervals-between-stations parameter is printed under assembly dividers.

**FIGURE 3.4.**

Labels display top depth, bottom depth and number of intervals-between-stations.

Breaks

This section describes how to define the structural members running from front to rear of a fuselage assembly, such as longerons, flooring and keel beams. It also describes the options for integrating structural components in the wing and fuselage. To access these options, select *breaks* from the Edit Mode menu and then use the mouse to select which assembly or assemblies to edit. (See table 3.4.) Note that only the cyan-colored assembly dividers can be selected.

Mouse Button	Selects	Usage
left	one or more assemblies	select first frame of starting assembly and last frame of ending assembly

TABLE 3.5. Mouse usage for selecting one or more assemblies

Mouse Button	Selects	Usage
middle	one assembly	select first frame
right	all assemblies	move mouse cursor into View Window and click button

TABLE 3.5. Mouse usage for selecting one or more assemblies

A picture of the “breaks” screen is shown in figure 3.5. Cross sections are drawn in the right View Window, one for each station in the assembly. The cross section for deep frames (stations with depth greater than zero) project inwards. The bottom four slider bars are used to adjust the angle and position of the cutting plane. The cutting plane appears as a cyan rectangle in the right View Window. In general, breaks are added by positioning the cutting plane and then selecting an option from the Breaks menu. A break is added only if the cutting plane intersects ALL the cross sections.

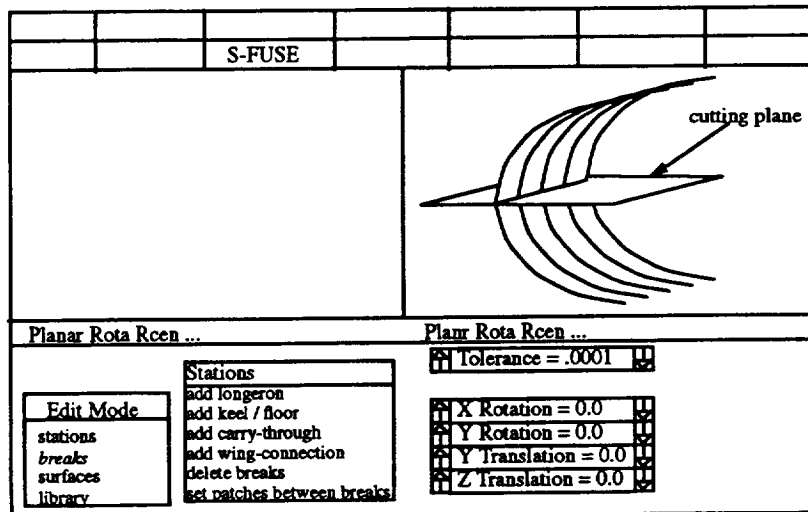


FIGURE 3.5. Illustration of screen for defining breaks.

add longeron

To add a longeron, position the cutting plane and select *add longeron* from the Breaks menu. A longeron is added where the cutting plane intersects the displayed cross sections. The longeron is displayed as a white line. If the cutting plane does not intersect one or more of the cross-sections, no longeron is added and you will get an error message that says “warning: longeron NOT added.”

add keel / floor

To add a keel or floor, position the cutting plane, select *add keel / floor* from the Breaks menu, and enter a patch-precision (range is 1-9.). The depth and direction of the floor or keel is deter-

mined by the orientation of the cutting plane. The patch-precision specifies how many rows of patches to fit between the outer and inner edge of the floor or keel. Figure 3.6 shows a beam with the patch-precision set to 3. For example, this sequence of actions adds a floor and a keel:

1. Set X Rotation and Y Translation to 0.
(Note: Y Translation set to 0 creates a floor that extends to the center line, set to a negative number creates a floor that extends past the center line, and set to a positive number creates a floor that is short of the center line.)
2. Move cutting plane to desired floor position by adjusting Z-translation.
3. To add floor, select add floor / keel and enter a number for the patch-precision.
4. Set X Rotation to -90.
5. To add keel, select add floor / keel and enter a number for the patch-precision.

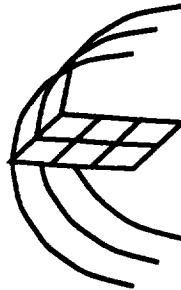


FIGURE 3.6. Beam with the patch-precision set to 3.

add carry-through

To create carry-through structure, you need to have input at least one root rib when you entered S-FUSE mode. The assembly must also satisfy these three constraints:

1. There is a station defined for each spar;
2. The stations corresponding to the leading edge and trailing edge spars are assembly dividers; and
3. All stations in the assembly are deep frames or none of them are.

When you select *add carry-through* from the Breaks menu, SMART adds two breaks to each of the cross sections. One break is the where the cross section is intersected by the upper wing surface. The other is where it is intersected by the lower wing surface.

add wing-connection

Whereas carry-through structure is continuous through the fuselage, the *add wing-connection* option defines spars that attach to the fuselage frames.

delete breaks

Select *delete* from the Breaks menu. The left mouse button is used for selecting the break. Going from top to bottom, each click of the left mouse button cycles you to the next break. The middle mouse button deletes the selected break. When you are finished deleting unwanted breaks, click the right mouse button.

set patches-between-breaks

This option increases the number of patches that are created when you generate the structural fuselage. The mouse buttons work the same way as *delete breaks*. When you click the middle mouse button, SMART prompts you to enter a number of patch rows to uniformly distribute between the selected breaks. The default number of rows is one.

if you get an error message...

In general, if SMART fails to add a break it is because either (1) one or more of the cross sections were NOT intersected by the cutting plane; or (2) the tolerance¹ used by the intersection algorithm was set to too small a number. First, check whether you need to adjust the cutting plane. Use the PLNR View Option to look at the ring frames from different perspectives. Sometimes although the cutting plane appears to intersect a frame in a side view, it is obvious that it doesn't in a front view. If the cutting plane appears to intersect the frames in all three of the planar views, then use the top slider bar to adjust the intersection tolerance.

1. To create a break, the program calls a patch-curve intersection algorithm to determine where the cutting plane intersects each cross section. The algorithm recursively subdivides the patch and curve until the algorithm either converges on the intersection point or exceeds a maximum number of iterations. Changing the tolerance affects the patch-curve intersection algorithm in two ways. First, tolerance is a factor in determining the maximum number of iterations. A smaller tolerance yields a higher maximum. Second, the tolerance is used for comparing a points on the patch to points on the curve. The algorithm succeeds when the curve-point and the patch-point match within the given tolerance.

Surfaces

To access the options for generating the structural surface, select *surfaces* from the Edit Mode menu. Figure 3.7 shows the three menus that are displayed.

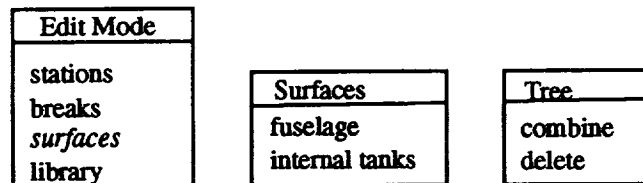


FIGURE 3.7. Menus for generating the structural fuselage.

fuselage

To generate a structural model, select *fuselage* from the Surfaces menu. Next, specify which assembly to generate by selecting an assembly-divider with the left mouse button or specify the entire fuselage by clicking the right mouse button anywhere in the right View Window. SMART resurfaces the fuselage so that the patch boundaries coincide with the underlying structural members that you have defined.

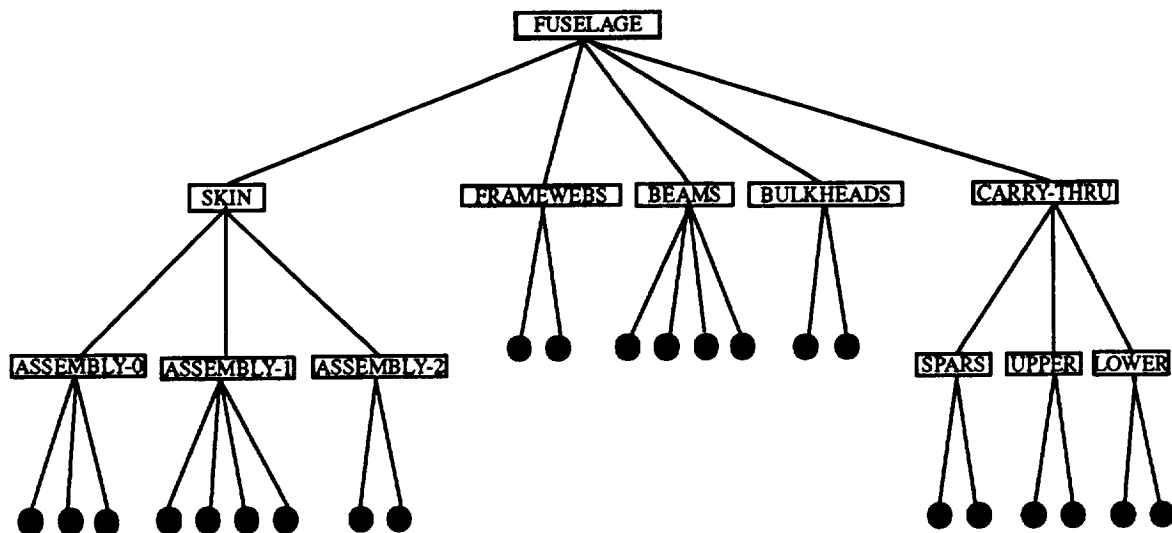


FIGURE 3.8. Typical component hierarchy.

The surface that is generated is stored in the tree. Figure 3.8 shows a typical component hierarchy. The circles in the figure represent the individual components. Although not shown in the figure, the first part of a component's name identifies which assembly it belongs to and the second part specifies

the type of component. Components for the fuselage's outer surface, beams, bulkheads, ring frame webs and wing integration are organized into different groups. (see table)))

Fuselage Group	Description
SKIN	The fuselage's outer surface is grouped by assembly. Each assembly is made up of one or more segments. A segment is the surface between two "breaks".
BEAMS	Each beam within the fuselage is a separate components. If a beam extends across more than one assembly, then a component is created for each assembly.
FRAMEWEBS	Components contain surfaces for all the deep frames in an assembly.
BULKHEADS	Each bulkhead is entered as a separate component.
CARRY-THRU	A group is created for each wing. Components are organized into four sub-groups: spar webs, upper surface, lower surface and ribs. If the wing spans more than one assembly, then separate components are generated for each assembly.

TABLE 3.6. Organization of the component hierarchy.

wing-fuselage integration

If carry-through or wing-connection is defined, SMART creates structure that integrates structural components in the wing and fuselage. Before generating carry-through, SMART asks "Vertical Drop?". If you answer no, SMART generates spar webs that are like the left-hand of figure 3.9. The diagonal that connects the upper wing break to the lower wing break divides the spar web into two patches. If you answer yes, your spar webs look more like the right-hand side of figure 3.9. Two vertical lines, one dropped from the upper wing break and another coming up from the lower wing break, divide the spar web into three patches.

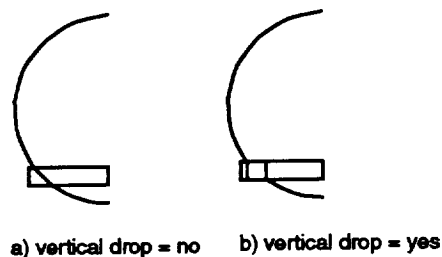


FIGURE 3.9. Vertical drop option for wing carry-through.

Note: An extraneous surface is generated in the X-Z plane when the wing is a vertical tail. To correct this problem, you can enter SMART's Tree mode and delete the nodes which contain the lower surface. For example, you would delete "LOWER" from the tree illustrated in figure 3.8.

internal tanks

Selecting *internal tanks* from the Surfaces menu generates surfaces for all dome caps and tank barrels that were entered as input to S-FUSE. The surfaces that are generated are stored in a two-level tree. The surface of each dome and barrel is stored in a separate node.

combine

The *combine* option consolidates the geometry of all the objects in a group node into a single geometry node. This option is useful if the surfacing algorithm creates objects whose individual parts you do not need to edit. This option also helps reduce the size of the tree. To combine, select *combine* from the Tree menu and then select the group node from the Tree.

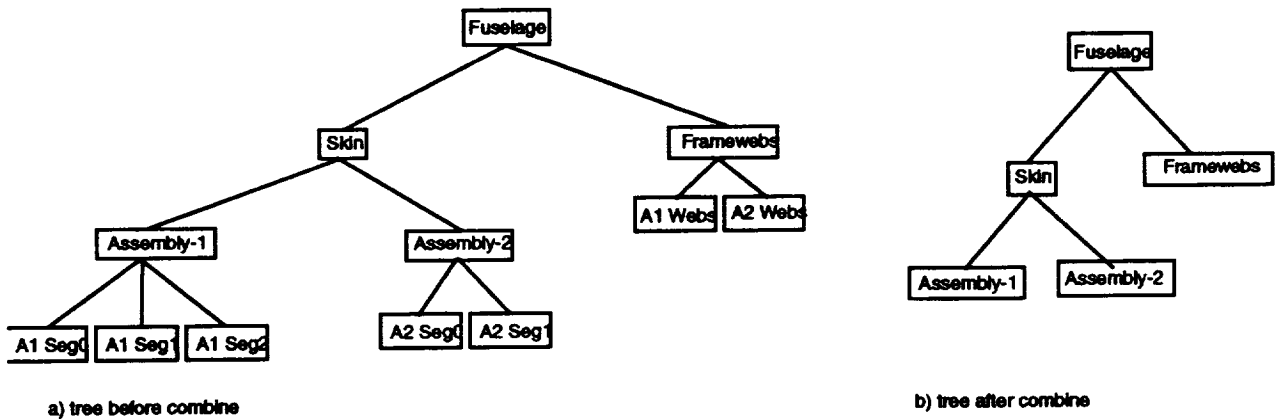


FIGURE 3.10.

The *combine* Option is applied to "Assembly-1", "Assembly-2", and "Framework" nodes.

delete

If you re-generate the structural surface, the old surface is not overwritten. It remains in the Tree unless you explicitly delete it. To delete, select *delete* from the Tree menu and then select the node to be deleted from the Tree.

ct rib

After a model has been generated, you can add one or more ribs to the carry-through structure. (See fig. 3.11.) Select *ct rib* from the Tree menu and then select the group node containing the carry-through structure. To add a rib, use the slider bars to position the cutting plane (plane must intersect both the upper and the lower surface) and select *add rib* from the CT Rib menu. To exit *ct rib* mode, select *done*.

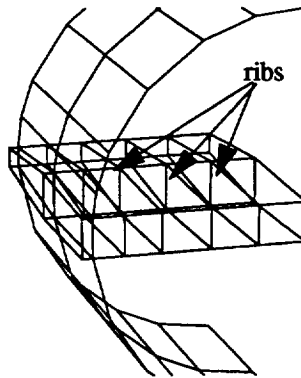


FIGURE 3.11. Illustration of rib added to carry-through structure.

Library

You can save your work in a file. These files have a “.sfuse” extension and are stored in your working SMART directory. Operations for reading, writing and deleting “.sfuse” files are available when you select *library* from the Edit Mode menu. As shown in figure 3.12, you are provided with a menu of file operations.

Edit Mode	Library
stations	read
breaks	save
surfaces	replace
library	delete

FIGURE 3.12. Menus for writing or reading the structural fuselage to a file.

read

To read a file, select *read* from the File Options menu and select a file to read from the File Chooser.

save

To write a file, select *save* from the File Options menu. At the prompts, type in a file name and a brief file description (80 characters or less). SMART writes the current structural fuselage definition to the file.

replace

The *replace* option is similar to the save option except that you chose an existing file.

delete

The *delete* option lets you remove files.

CHAPTER 4

MERGE

This chapter describes MERGE Mode, which lets you improve the connectivity among individual patches in a structural geometry. To be consistent with FEA requirements, it is necessary that patches that border each other should share vertices along common edges. As shown in figure 4.1, the surfaces generated in S-FUSE and S-WING Modes do not always meet this constraint. Unmatched vertices typically occur at the juncture of two fuselage or wing assemblies, and where the structural wing joins the fuselage's carry-through structure.

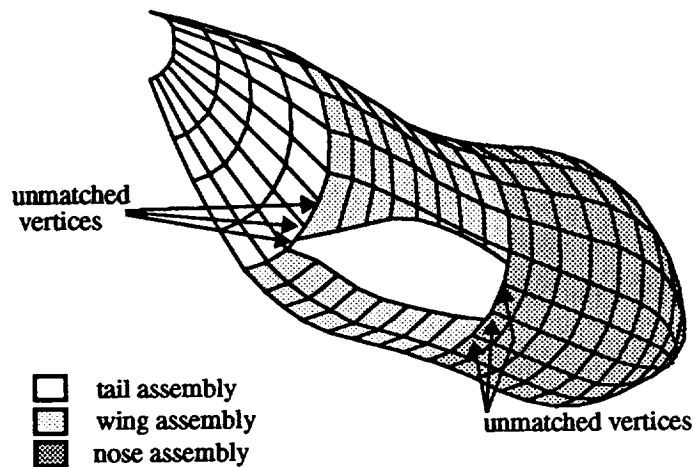


FIGURE 4.1. Unmatched vertices typically occur at the juncture of two fuselage or wing assemblies.

Table 4.1 lists options that you can use to fix surfaces that contain patches that do not share vertices along common edges.

Operation	Edit Mode	Description
merge fuselage	auto	Aligns patch vertices of structural fuselage.
merge wing	auto	Aligns patch vertices of structural wing.
merge wing-to-fuselage	auto	Aligns patch vertices of wing's root rib and fuselage's carry-through structure.
merge	manual	Creates triangular transition patches to align surface-1 with surface-2.
split	manual	Splits patches in surface-1 to align surface-1 with surface-2.

TABLE 4.1. Merge operations.

Operation	Edit Mode	Description
join	manual	Gets rid of unnecessary 3-sided patches
equivalence	manual	Sets patch edges in surface-1 to be equivalent to patch edges in surface-2.
outline patches		Creates curves to make wire-frame of surface.
undo		Restores state to just prior to last user operation.

TABLE 4.1. Merge operations.

Two Edit Modes: Automatic and Manual

Merge operations are categorized as either automatic or manual. Automatic operations search the structural components of a fuselage or wing looking for vertices that are not shared along common edges. The software automatically fixes any errors that are found. In contrast, manual operations let you hand-pick the components and the method that is used to fix errors. Most manual operations require you to pick a master surface and a slave surface. The software looks for patch edges that are shared by both the master and the slave. If a common edge does not share vertices, the software uses the chosen method (e.g., merge, split or join) to modify the slave surface so that its vertices match those of the master edge. Only the patches of the slave surface are changed. The master surface remains the same.

The Process

In general, start with the automatic merge options. Inspect afterwards for vertices that are not shared along common edges. If any are found, use the manual merge options to correct. If all else fails, you may need to use the PATCH TOOL options to manipulate individual patches in the surface (see chapter 9 of the SMART User's Guide). After the model is fully merged, use the *outline patches* option to extract curves for representing one-dimensional beam elements.

Caveat

The merge options ignore transformation matrices. You can use the reset option of PACKAGING Mode to make certain that all transformations have been applied to the geometry.

The Merge Options

To access the merge options, select **MERGE** from the main menu. The **MERGE Mode** menus are shown in figure 4.2.

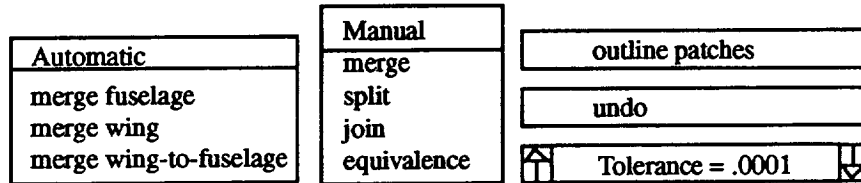


FIGURE 4.2. Menus for **MERGE Mode**.

merge fuselage

When you select the *merge fuselage* option, the textport prompts you to select the group that represents all or part of a structural fuselage. After you select, SMART looks for geometric components that share a common edge and attempts to align any unmatched vertices that are found.

merge wing

This is similar to the previous option except the group that you select is a structural wing.

merge wing to fuselage

When you select the *merge wing to fuselage* option, the textport prompts you to select the structural fuselage and structural wing from the tree. If needed, triangular transition patches are inserted to align the wing's root rib to the fuselage's carry-through structure.

merge

When you select the *merge* option, the textport prompts you to select two surfaces from the tree. The first is the master surface and the second is the slave. As shown in figure 4.3, triangular patches are created where vertices on the master surface are not matched by vertices on the adjoining patches of

the slave surface. Only the slave surface is changed. To align the surfaces, it is usually necessary to perform a second merge by reversing slave and master.

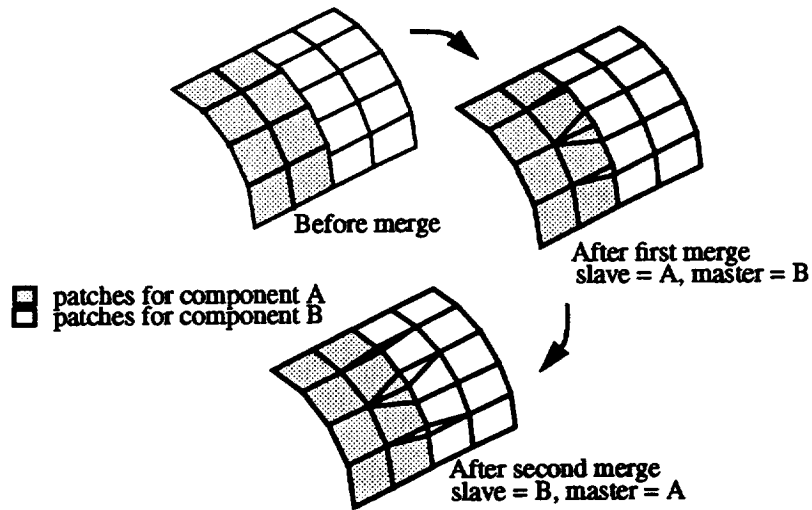


FIGURE 4.3. Illustration of the "merge" Option.

split

The *split* option works just like the *merge* option. The difference is that an unmatched vertex on the master surface is resolved by inserting a row of patches into the slave surface (see fig. 4.4).

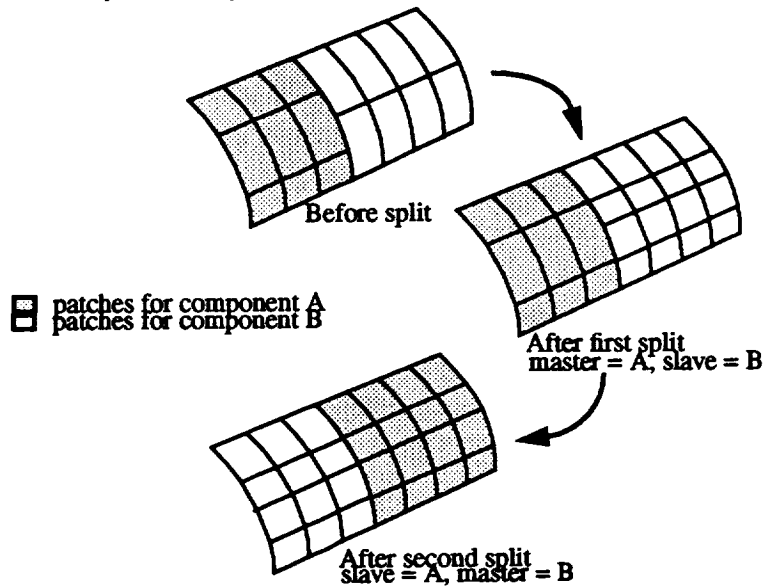


FIGURE 4.4. Illustration of the "split" Option.

join

When you select the *join* option, the textport prompts you to select a surface from the tree. As shown in figure 4.5, this option reduces the number of three-sided patches by joining the three-sided patches to patches that are adjacent to them.

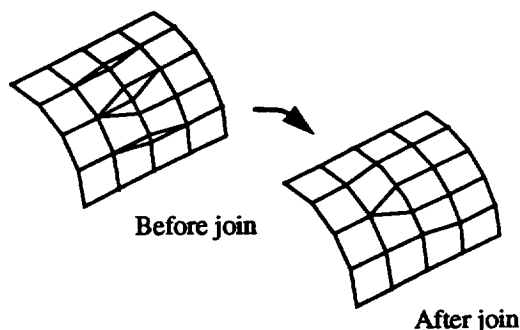


FIGURE 4.5. Illustration of the "join" Option.

equivalence

When you select the *equivalence* option, the textport prompts you to select a master surface and a slave surface from the tree. This option checks for edges that are common to both the master and the slave surface. As shown in figure 4.6, the slave edge is set equal to the master edge if the vertices on the slave edge are the same within a given tolerance as the vertices on the master. The purpose of this option is to resolve small inconsistencies in slope and endpoints between patches that share a common edge.

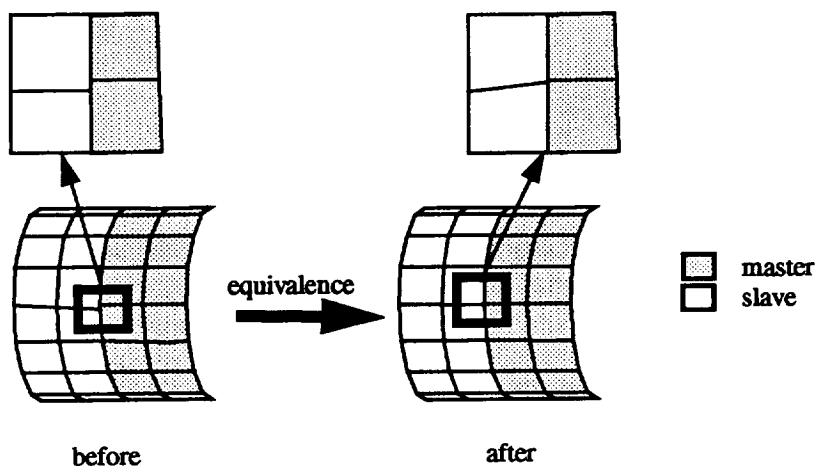


FIGURE 4.6. Illustration of the "equivalence" Option.

outline patches

The *outline patches* option adds two nodes to the tree. For fuselages, as shown in figure 4.7, the first node contains curves that are created by extracting the patch edges that run chord-wise in the surface and the second node contains curves that are created by extracting the patch edges that run span-wise. For wings, the first node contains curves that outline the spars and the second node contains curves that outline the ribs. When you select the **outline patches** option, the textport prompts you to select a surface from the tree and then enter names for the two new nodes.

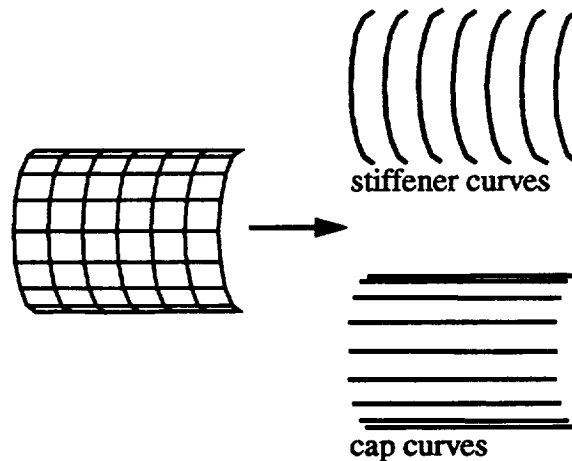


FIGURE 4.7. Illustration of the "outline" Option.

undo

The *undo* option restores all geometry in the tree to its state just prior to the last user operation.

Resetting the Tolerance

Point comparisons are ubiquitous in the algorithms used to merge patches. If a merge option fails to yield the results that you expect, it may be because the tolerance for coincident point resolution is set to too low or too high a number. You can adjust the tolerance by changing the value on the slider bar.

APPENDIX A

Bringing an Outside Model into S-WING

This appendix describes how to introduce a surface model into S-WING that was created on a system other than SMART.

Bringing in the Surface Model

Input to S-WING is an upper and lower wing surface. Symmetry across the X-Z plane is assumed. The surfaces should be continuous; that is, there should be no holes in the surface or gaps between patches. As shown in figure A.1, vertical tails and wings both require two surfaces. You need to provide an X-Z planar surface for the vertical tail because the program processes all wing-like objects the same way and does not distinguish between wings and vertical tails.

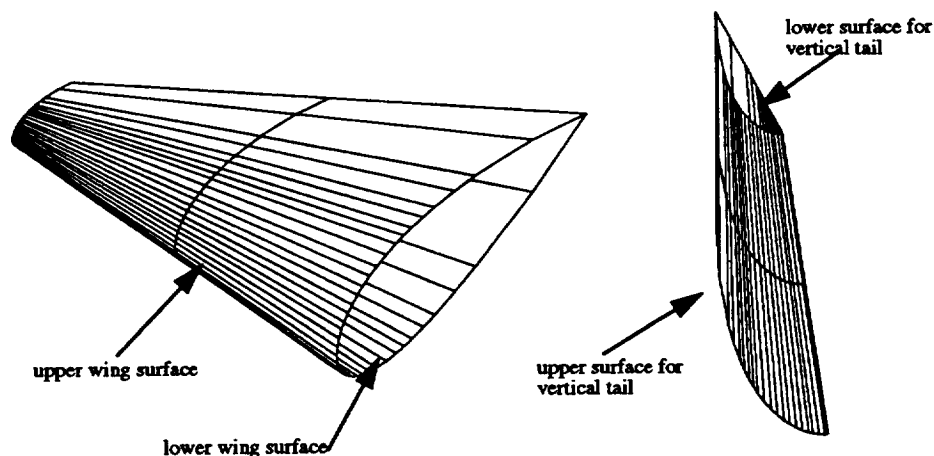


FIGURE A.1.

Example of surface model for wing (left) and for vertical tail (right.)

Figure A.2 shows the path that is followed when you bring a surface model into SMART as a file:

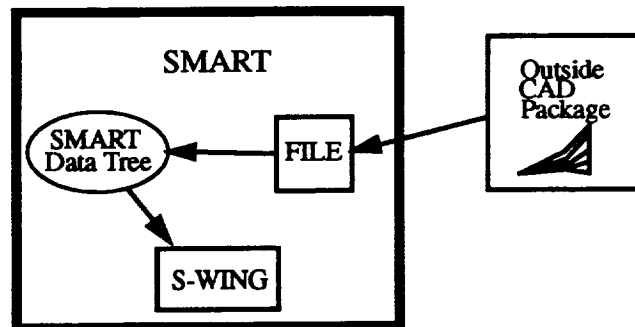


FIGURE A.2. Path for bringing outside model into structural wing editor.

The file formats that are available for reading in surfaces are SMART's own format and the PATRAN neutral format. For details on using these files, refer to chapter 12 of the SMART User's Guide.

When a file is read, depending on how the geometry is grouped together in the file, one or more nodes will be added to the Tree. It is required by S-WING that the surface model be represented either as one node containing the entire wing (patches for the upper wing surface must precede those for the lower) or as two nodes, one for each wing surface. If your geometry is not grouped in this manner, PATCH TOOLS Mode lets you move and copy patches from one node to another.

Selecting from the Tree

When you enter S-WING, you are prompted to select nodes from the tree representing the surface model. After each selection from the tree, you then select the appropriate entry from the Wing Geometry menu to indicate whether the node is an upper surface, a lower surface, or the entire wing.

Changing the Orientation

This section describes how to change the orientation of the original surface model. The objective is to orient the wing so that it is symmetric across the X-Z plane and is defined by the geometry on the positive side of the plane. Once the surface model has been selected from the Tree, it is displayed in the right View Window. The leading edge should be up and the tip to the right. If the wing is oriented correctly, click on *done*. Otherwise, use the menu that is provided to transform the surface model into the appropriate orientation:

- *XY, YX, XZ, ZX, YZ and ZY* rotates the wing into the indicated plane.
- *flip X* flips the wing-plane across the X-axis.

- *flip Y* flips the wing-plane across the Y-axis.

Moving the Root Rib Outside the Fuselage

This section describes how to move the root rib of the default planform. Once the surface model is correctly oriented, SMART automatically extracts a planform. Perpendicular chord-lines break the planform into one or more sections. By default, these break chords are stationed wherever slope discontinuity is detected along the leading or trailing edges of the surface model. The root chord of the planform's most inboard section is stationed at the minimum Y of the surface model. In most cases this is $Y = 0$ or the vehicle's center line. A problem occurs if you intend to later join the wing's structural model with that of the fuselage. Recall that a constraint for wing-fuselage integration, as shown in figure 1.15, is that the root rib of the wing's planform must lie outside the fuselage. To remedy this problem, SMART puts the mouse cursor into the right View Window and prompts you to select the fuselage from the Tree. Selection of a fuselage node translates the root rib so that it lies completely outside the selected geometry.

Editing the Planform

This section describes how to modify the default planform. This is the last step before structural editing begins. The planform editor that is invoked is the same one that is used for wing generation in PRIMITIVES Mode.

Planform display

The View Window displays the planform in white overlying the surface model which is drawn in blue. It is a fixed planar view, and all View Options except for Wind, X-ZM, and GOFF/GON are turned off. Occasionally, SMART fails to extract an outline of the wing from the surface model. In this case, exit S-WING and use the *switch S & T* option of PATCH TOOLS Mode to change the parametric orientation of the surface model.

Moving Planform Vertices

This section describes how to use the mouse cursor to move the vertices of the planform. As you move the mouse cursor into the View Window, the cursor changes from a cross hair to a small white square. The planform vertices are also marked by small squares. To lock onto a vertex, move the mouse cursor over it and press the left mouse button. As long as the left mouse button remains depressed, you can move the vertex vertically by moving the cursor up and down. You can also move

break chords as a whole. Selecting either endpoint of the desired chord with the middle mouse button lets you move the chord vertically. Selection with the right mouse button moves the chord horizontally.

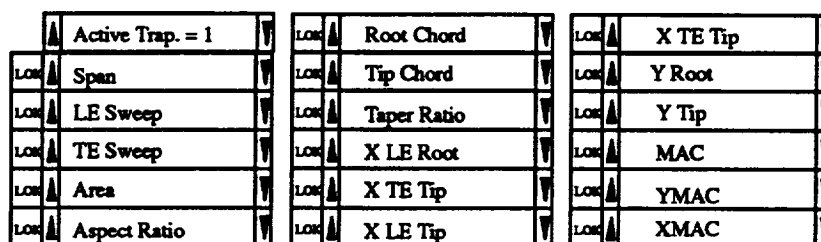


FIGURE A.3. Slider bars for editing planform.

Using the Slider bars

This section describes the 18 slider bars that SMART provides for adjusting planform parameters. These slider bars are shown in figure A.2. Slider bars modify only one planform section at a time. The *Active Trap* slider bar lets you select the planform section that you want to modify. In the View Window, SMART highlights the activated section by drawing it in yellow.

Some parameters are interdependent. Changing the value of one parameter causes all dependent parameters to also be updated. If you want a parameter to stay constant, click on the *lok* toggle of its slider bar. This locks in the current value of the parameter and prevents it from being inadvertently changed as you edit other parameters. The function of each slider bar is as follows:

- *Active Trap* specifies which section of the planform to edit.
- *Span* adjusts the wing 50% span.
- *LE Sweep* and *TE Sweep* adjusts the leading and trailing edge sweep.
- *Area* adjusts the area.
- *Aspect Ratio* adjusts the aspect ratio.
- *Root chord* and *Tip Chord* adjust the root and tip chord.
- *Taper Ratio* adjusts the taper ratio.
- *X LE Root*, *X TE Root*, *X LE Tip*, and *X TE Tip* assigns the X-coordinates of the four vertices of the active section.

- *Yroot* and *Ytip* assigns the Y-position of the root and tip rib. The *Yroot* slider bar provides an alternative method for moving outwards the root rib of the default planform.

SPLIT	APPEND	LIBRARY	AUTO WIN	UNDO
COMBINE	DELETE	FLAPS	REFLECT	QUIT

FIGURE A.4. Menu panel for editing planform.

Using the Menus

A panel (see fig. A.3.) which is above the slider bars contains these menu items:

- *SPLIT* splits the current section into two sections. The split line is a new break chord that is added at a user-defined percentage of the active section's span.
- *COMBINE* combines two sections into one. The active section is joined to the section that precedes it.
- *DELETE* deletes the root chord of the active section. The root chord appears highlighted in red in the View Window.
- *LIBRARY* lets you save the current planform to a file or read a new planform from a file.
- *AUTO WIN* turns the auto windowing feature on and off.
- *REFLECT* turns reflection of the planform across the root on and off.
- *UNDO* restores the planform to its state just prior to the last operation.
- *QUIT* exits the planform editor. Before you exit, check that none of the planform's vertices lie outside the XY-projection of the surface model. Quitting the planform editor causes the S-WING menus to appear on the screen.

aerowing is Added to the Tree

A group named AeroWing is added to the tree by the S-WING editor. It is located to the right of any structural models that are generated. The AeroWing group contains the surface model and its planform. Any transformations that were applied in orienting the original surface are also captured. If later you wish to re-edit the surface model, select AeroWing from the tree. The S-WING menus come up directly. If you do not want to by-pass the planform editor, pick instead the geometry node that is the child of AeroWing.

APPENDIX B

Constraints on Geometric Input to SFUSE

This appendix describes constraints on geometric input to S-FUSE.

Fuselage

The surface of the input fuselage must satisfy these constraints:

- The surface is represented by Bezier patches.
Note that SMART automatically converts any surface that is read from a file to a Bezier representation.
- The surface is continuous (there are no holes in the surface or gaps between individual patches.)
- The center line is at $Y=0$. The body length is oriented along the X-axis.

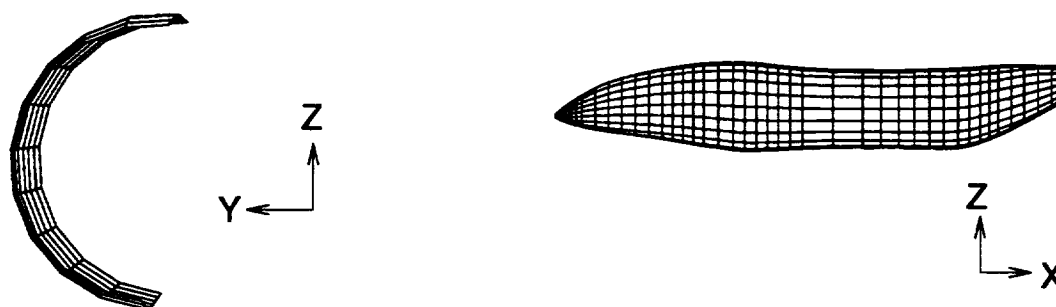


FIGURE B.1.

In general, a reflected surface represents the fuselage body.

- It is desirable that the surface be symmetric across the X-Z plane and be defined by the geometry on the positive side of the plane (see fig. B.1.)

This constraint is not mandatory. However, S-FUSE was originally designed to accept only reflected surfaces and runs best when this type of input is used. Lesson 2 of Appendix C, a tutorial on wing/tip fin integration, illustrates a case where it is necessary to bring in an un-reflected surface model.

- Adjacent patches share vertices along common edges. (See fig. B.2.)

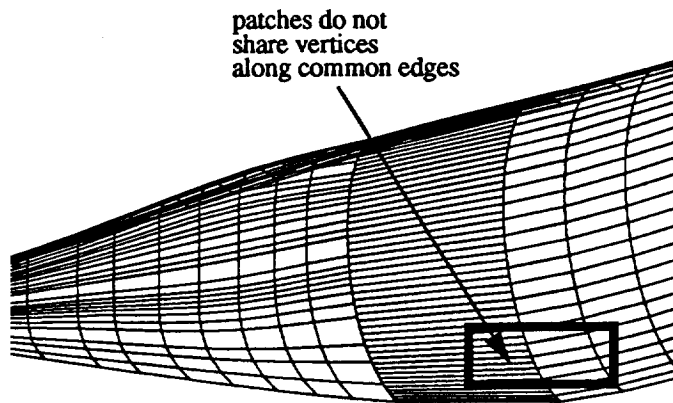


FIGURE B.2.

Example of bad input geometry.

Root Rib

Wing-fuselage integration requires you to also bring in the root rib of the structural wing. Figure B.3 shows geometry defining a root rib:

- The root rib is represented by Bezier patches.
- The surface is continuous (there are no gaps between individual patches.)
- Left and right patch edges define where spars intersect the root rib.
- Top and bottom patch edges define the upper and lower curve of the root rib.
- Patches do not have degenerate edges.

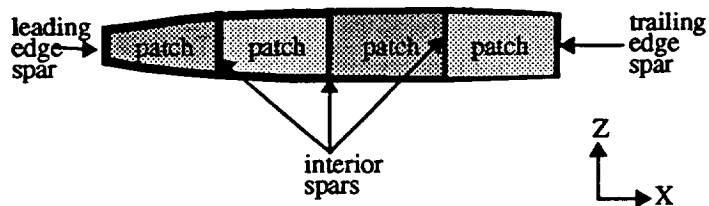


FIGURE B.3.

Constraints on root rib's geometry.

There is an additional requirement if the root rib is for a vertical tail. In this case, the root rib must be symmetric across the X-Z plane. Only the half of the root rib which is on the positive side of the X-Z plane is introduced into S-FUSE. (See fig. B.4.)

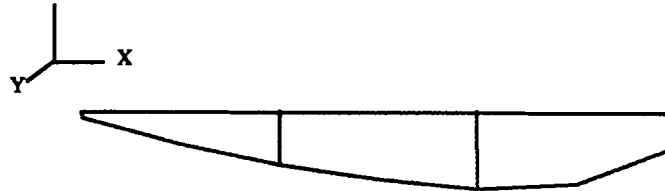


FIGURE B.4. Root rib for vertical tail.

Root ribs that are generated by SMART's structural wing editor satisfy these constraints. As shown in figure B.5, you can select either the actual node containing the wing body's root rib (default label is bRoot) or the root node of the structural wing model.

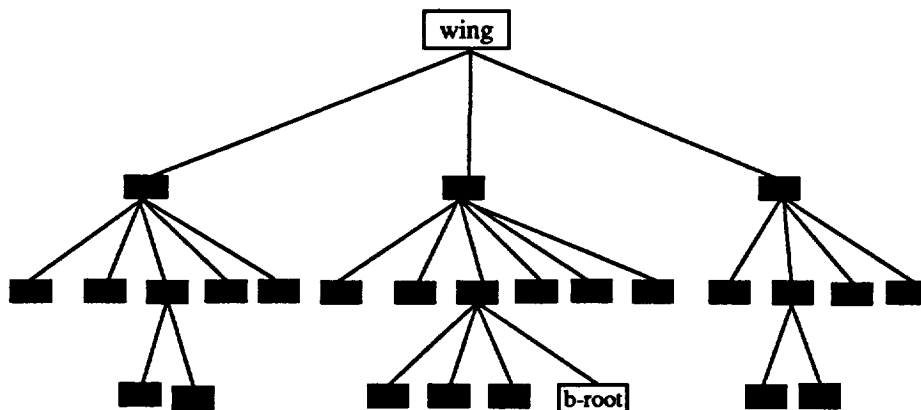


FIGURE B.5. Illustration of tree showing position of root rib component of wing body section.

A constraint for wing-fuselage integration, as shown in figure 1.12, is that the root rib of the wing must lie outside the fuselage. For more information, refer to the section of Appendix A that is titled "Moving the Root Rib Outside the Fuselage".

Tank

This section describes constraints on the input that defines tanks that are internal to the fuselage (e.g., a fuel tank). Tank geometries should satisfy the first four of the constraints which are listed for the

fuselage. Barrels and dome caps are selected individually from the tree. Domes can be selected without specifying a barrel or vice versa.

If your tank was created by the *tank* option of PRIMITIVES, the length of the tank lies along the Z-axis and the patches for the barrel and both the dome caps are lumped together in a single node. Before entering the S-FUSE you need to:

- Enter PACKAGING Mode. Rotate the tank so that the length of the tank lies along the X-axis. This is achieved by rotating the tank -90 degrees in both X and Y. Additional translation and scaling transformations can be applied in order to orient the tank with respect to the fuselage, and then reset tree.
- Enter PATCH TOOLS Mode. Select patches that are on the negative side of the X-Z plane. Select *delete* to remove patches from the tank.
- Select patches for dome cap positioned on the positive Z-axis. Select *move* to take patches from the tank's node and place them into another node (i.e., a node called "left_dome"), and then repeat for dome that is on the negative X-axis.

For details on using these options, refer to the SMART User's Guide.

APPENDIX C**Tutorial**

This appendix contains recipes for generating structural models for two different vehicles. The objective is to guide the novice user through the process of creating a structural model.

Before You Begin

Before you begin, make sure that you have the following items:

- Access to SMART (version 3.0 or higher) on your account.
- Familiarity with the basic concepts covered in the SMART User's Guide and the SMART Structures User's Guide.
- Tutorial files installed in the directory that you use for reading and writing SMART data files. These files can be copied from the SMART *program_data/sf_tutorial* directory. Table C.1 lists the files.

file name	file type	description
v1-SurfModel.s	SMART file	input geometry for first vehicle (lesson 1)
v1-mw.fem	S-WING file	defines rib and spar pattern for first vehicle's main wing
v1-vt.fem	S-WING file	defines rib and spar pattern for first vehicle's vertical tail
v1-fuse.sfuse	S-FUSE file	defines structure for first vehicle's fuselage
v1-StrcModel.s	SMART file	structural model for first vehicle
v2-SurfModel.s	SMART file	input geometry for second vehicle (lesson 2)
v2-fuse.sfuse	S-FUSE file	defines structural layout for second vehicle's fuselage
v2-StrcModel.s	SMART file	structural model for first vehicle

Lesson 1

Overview

Lesson one shows you how to generate structure for a vehicle that is composed of a main wing, a vertical tail and a fuselage. After you complete this lesson, you should be able to:

- Generate a structural model for a wing.
- Generate a structural model for a fuselage.
- Merge structural components so that patches that border each other share vertices along common edges.

Figure C.1 diagrams the modeling approach of lesson one. First, the surface model of the vehicle is read into SMART. Next, structural models are generated for the main wing, the vertical tail and the fuselage. The final step merges the patch meshes that are generated for the structural components.

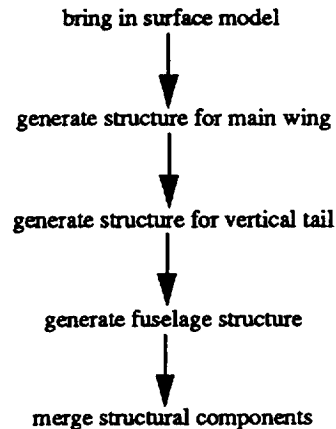


FIGURE C.1. Modeling approach for lesson 1.

Bring in the Surface Model

1. Enter FILE Mode. Read SMART file called *v1-SurfModel*. The geometry for the surface model is grouped into three nodes: *main wing*, *v tail* and *fuselage*.

Generate the Structural Model for the Main Wing

2. Select S-WING from the main menu.
3. Select the node called *main wing* from the Tree. Since this node contains patches for both the upper and lower surface, select *entire wing* from the Wing Geometry menu.
4. The surface model is displayed on the screen. You can tell the wing is oriented correctly because the leading edge is up and the tip is to the right. Click on *done* to confirm that the orientation is correct.
5. Select *fuselage* from the Tree. This moves the root rib outside of the fuselage, a constraint that is required later for wing-fuselage integration.
6. The planform of the wing is displayed in the View Window. Below the View Window are an assortment of slider bars for editing the planform. You do not need to edit the planform. Select *QUIT* from the panel that is located above the slider bars. Quitting the planform editor causes the S-WING menus to appear on the screen.

As shown in figure C.2, the planform that appears in the right View Window has an inboard section and an outboard section. The leading edge and trailing edge spars of both sections are coincident with the edges of the planform.



FIGURE C.2.

Illustration of screen after step 6 of lesson 1.

7. Place the leading edge spar (inboard and outboard wing sections):
 - Verify that *wing box* is highlighted on the Edit Mode menu, *leading* is highlighted on the Edge menu, *skew* is highlighted on the Orientation menu, and *Active Sect.* is set to 1.
 - Set *Root Chord%* to 17 and *Tip Chord%* to 20. Notice that because the software provides for automatic matching at the common chord, the leading edge spar of both wing sections is updated.
8. Place the trailing edge spar (inboard wing section):
 - Select *trailing* from the Edge menu and *parallel* from the Orientation menu.
 - Set the *Chord%* to 97.
9. Place the trailing edge spar (outboard wing section):
 - Set *Active Sect.* to 2.
 - Select *skew* from the Orientation menu.
 - Decrease *Tip Chord* percentage until the spar is parallel to the trailing edge. Before continuing to step 10, your screen should look like figure C.3.



FIGURE C.3.

Illustration of screen after step 9 of lesson 1.

10. Enter 4 vertical ribs (wing body region of the inboard wing section):

- Select *ribs* from the Edit Mode menu, *wing body* from the Box menu and *equally spaced* from the Create menu.
 - Set *Active Sect.* to 1.
 - Set *No. Ribs* to 4.
11. Extend ribs into the leading edge region and trailing edge region:
- Select *leading edge* from the Box menu and set *No. Ribs* to 4.
 - Select *trailing edge* from the Box menu and set *No. Ribs* to 4.
12. Enter 5 equally spaced ribs in the outboard wing section. (See fig. C.4.)



FIGURE C.4.

Illustration of screen after step 12 of lesson 1.

13. Enter two spars (outboard wing section):
- Select *spars* from the Edit Mode menu, *wing body* from the Box menu and *equally spaced* from the Create menu.
 - Set *Active Sect.* to 2.
 - Set *No. Spars* to 2.

(See fig. C.5.)



FIGURE C.5.

Illustration of screen after step 13 of lesson 1.

14. Extend spars into the inboard wing section:
- Set *Active Sect.* to 1.
 - Verify that *wing body* is highlighted in the Box menu and select *match adjacent* from the Create menu.

(See fig. C.6.)



FIGURE C.6.

Illustration of screen after step 14 of lesson 1.

15. Move endpoints of inboard spars so that they are horizontal:

- Select *move* from the Modify Menu.
- Move the mouse cursor into the right View Window, position the mouse cursor where bottom spar intersects root rib, and press left mouse button.

Note: Small squares mark the spar endpoints. If the square is yellow, it means that the endpoint is shared by more than one rib or spar. For example, the yellow squares on the tip rib indicate that the spars of the first assembly match up with the spars of the second assembly.
- Drag the endpoint so that spar is approximately horizontal, and then release mouse button.
- Repeat for top spar.

(See fig. C.7.)



FIGURE C.7.

Illustration of screen after step 15 of lesson 1.

16. Add spars to inboard section:

- Select *perpendicular to ribs* from the Orientation Menu.
- Move the cursor into the right View Window. Left-click on DO ALL.

(See fig. C.8.)

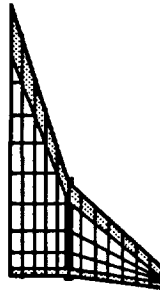


FIGURE C.8.

Illustration of screen after step 16 of lesson 1.

17. To generate the structural model:

- Select *surfaces* from the Edit Mode Menu and *entire wing* from the Surfaces Menu.
- After the prompts, enter a name (e.g. “main-wing”) and a two-letter identifier (e.g. “mw”).

S-WING generates a structural model that is organized as a hierarchy of components. The name that you entered is given to the top node of the hierarchy. Names for the other nodes are created by stringing together the user-supplied identifier, a system-supplied component label, and the wing section number. For example, the root rib in the wing body of the inboard wing section would be named “mw_broot_1”.

- The resulting geometry, as shown in figure C.9, is displayed in the left View Window.

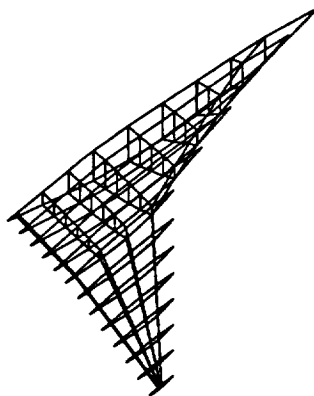


FIGURE C.9.

Structural model for main wing.

Generate the Structural Model for the Vertical Tail

18. Bring vertical tail into S-WING:

- Select **S-WING** from the main menu.
- Select the node that is named *v_tail* from the tree and then *entire wing* from the **Wing Geometry** menu.
- Use the *xz* and *flip x* options to orient the wing. When the orientation is correct, click on *done*.
- Don't select a fuselage from the tree. Move the cursor outside the right View Window. Note: The root rib of the vertical tail is already external to the fuselage.
- Select **QUIT** to exit the planform editor.

19. Position the leading and trailing edge spar:

- Verify that *wing box* is highlighted on the **Edit Mode** menu. Select *leading* from the **Edge** menu and *parallel* from the **Orientation** menu. Set the *Chord%* to 20.
- Select *trailing* from the **Edge** menu and *parallel* from the **Orientation** menu. Set *Chord%* to 85.

(see fig. C.10.)

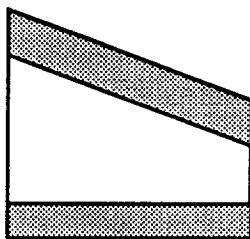


FIGURE C.10.

Illustration of screen after step 19 of lesson 1.

20. Add ribs and spars:

- Select *ribs* from the **Edit Mode** menu, *wing body* from the **Box** menu, and *equally spaced* from the **Create** menu. Set *No. Ribs* to 2.
- Select *spars* from the **Edit Mode** menu and *equally spaced* from the **Create** menu. Set *No. Spars* to 1.

(see fig. C.11.)

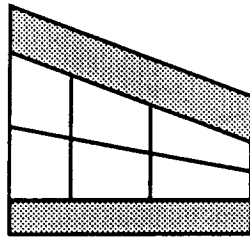


FIGURE C.11. Illustration of screen after step 20 of lesson 1.

21. Generate the structural model.
22. Enter PACKAGING mode and rotate the structural model 90 degrees about the X-axis. (See fig. C.12.)

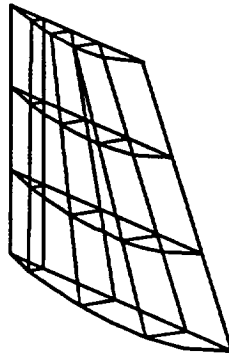


FIGURE C.12. Structural model of vertical tail after rotation.

23. Delete extraneous surface that was generated in the X-Z plane.
 - Enter TREE Mode.

- Delete *vt_tLower0*, *vt_bLower0*, and *vt_lLower0* from the tree. In figure C.13, these nodes are shown in reverse video.

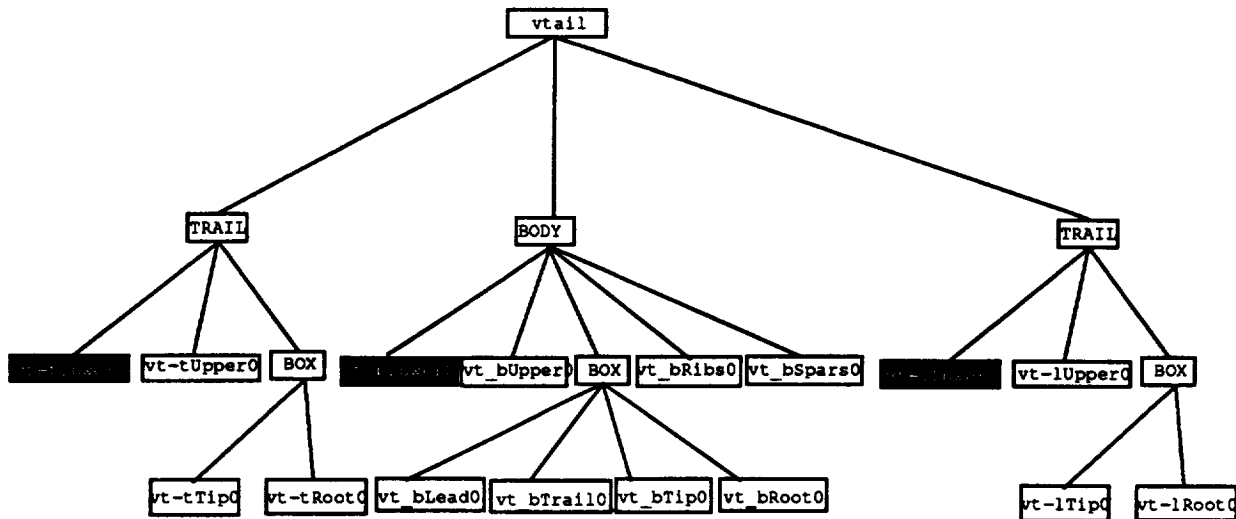


FIGURE C.13.

Illustration of tree for vertical tail. Delete nodes shown in reverse video to get rid of unnecessary surface in X-Z plane.

Generate the Structural Model for the Fuselage

24. Select S-FUSE from the main menu.
25. Select *fuselage* from the tree.
26. Select *main-wing* from tree.

SMART creates an upper and a lower surface by extruding the root rib to the fuselage's center line. The surfaces are displayed in the View Windows. You can use the View Options Menu to look at the extruded root rib from different angles. To accept, select *yes* from the Verify menu.

27. Repeat step 25 for the vertical tail.

Note: There are two options for generating wing-fuselage integration. The carry-through option will be used to generate structure for the main wing. This option requires that the root rib is extruded to the center line. The frame-attachment option will be used for the vertical tail and it is only necessary that the extruded root rib penetrates the fuselage surface.

28. Move mouse cursor outside the View Window.

The fuselage is displayed as an outline of its X-Z projected area. As shown in figure C.14, stations have been added fore, aft, and at each spar location. Below each station is an L-shaped line called its handle.

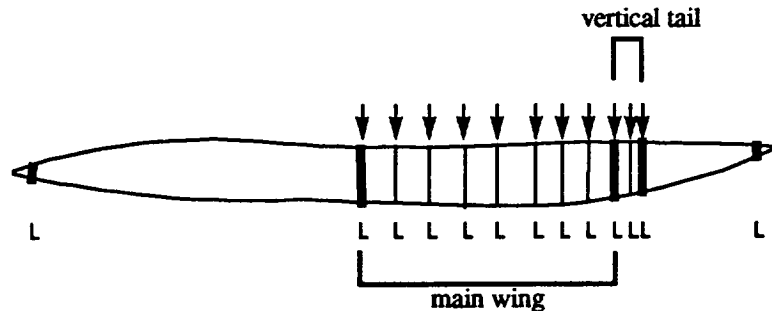


FIGURE C.14. Illustration of screen after step 27 of lesson 1.

Move the mouse cursor over a handle. Notice that the X-coordinate of the station is displayed in the screen area above the right View Window. A station is selected by clicking on its handle.

Note: The program displays a station's X-coordinate to three decimal places. However, for the sake of brevity, this tutorial truncates the value of a station's X-coordinate. Thus, "Select station at X= 136" really means "Select station at X=136.909".

Another thing to keep in mind is that while it is OK to add more stations to a wing assembly, you do not want to move or delete the stations that are under the green arrows. These arrows show where the wings' spar components will be joined to the fuselage's ringframes. (See chapter 2 for a list of constraints regarding wing-fuselage integration.) If you accidentally move one of these stations, it is easy to move it back to its original position. Spar locations and any other location denoted by an arrow are called "sticky points". When the mouse cursor is close to a sticky point, the cursor automatically resets to the exact value of the sticky point.

29. Distribute stations:

- Select *distribute by distance* from the Stations menu.
- Select start station at X = 3 (i.e., left-click on the station handle.)
- Select stop station at X = 136.
- Enter 7 for the unit distance. A station is added every seven units.
- Select start station at X = 247 and stop station at X = 297. Enter 7 for the unit distance.

(See fig. C.15.)

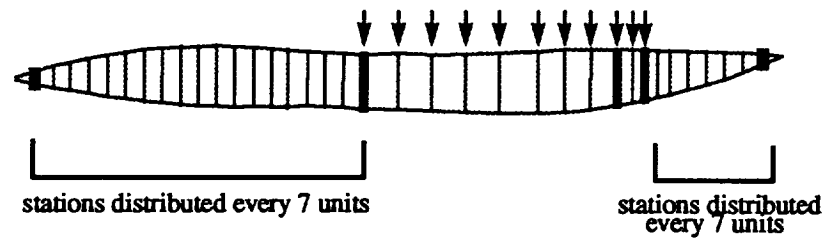


FIGURE C.15. Illustration of screen after step 28 of lesson 1.

30. Subdivide left-most assembly:

- Select *toggle assembly divider* from the Stations menu.
- Select station at X=31.

(See fig. C.16.)

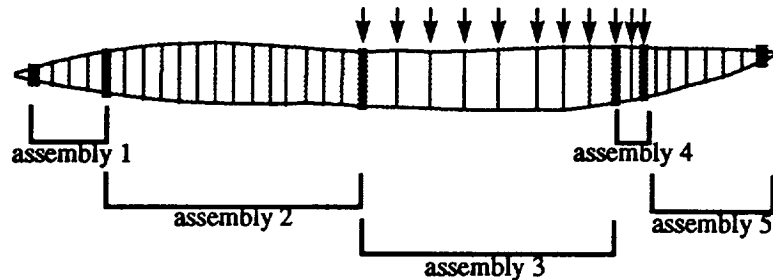


FIGURE C.16. Illustration of screen after step 29 of lesson 1.

31. Add bulkheads:

- Select *toggle bulkhead* from the Stations menu.
- Select stations at X=31, X=38 and X=247.

(See fig. C.17.)

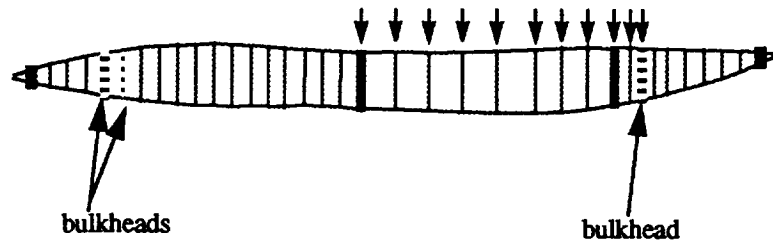


FIGURE C.17. Illustration of screen after step 30 of lesson 1.

32. Assign depths for creating ringframe webs:

- Select *display labels*.

Station depths are displayed on the screen. The depths are hard to read if stations are bunched close together. To remedy this problem you can use the X-ZM view option to zoom and pan the fuselage or 1-VU to increase the size of the right View Window.

- Set both slider bars to $<.1>$.

Two sliders bars are provided because depth may vary linearly from the top to the bottom of the frame. Setting both slider bars to same value creates frame webs of constant depth.

- Select *set multiple* from Depth menu.
- Select start station at X=38 and stop station at X = 247.
- Depths are assigned to the selected stations and the stations that are between them.

(See fig. C.18.)

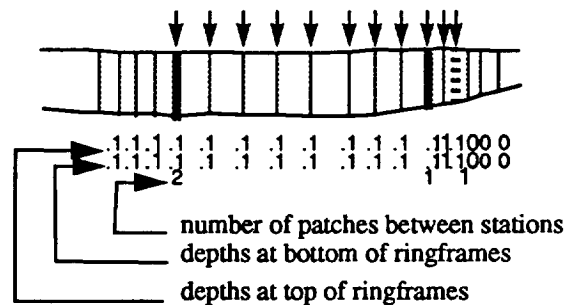


FIGURE C.18. Illustration of screen when *display labels* is turned on.

33. Select *set intervals between stations* from the Stations menu. Select cyan-colored station at X = 136. At the prompt, enter 2.

Later, when you generate the structural model, more patches will be generated for this assembly than you might first expect. The software ordinarily generates one column of patches to represent the surface between two stations. However, setting the intervals-between-stations to 2 tells the software to generate 2 columns of patches between stations within this assembly.

34. Add carry-through break (integrates main wing to fuselage) to assembly 3:
 - Select *breaks* from the Edit Mode menu.
 - Left-click on first and last station of assembly 3 (stations at X = 136 and X = 234).
 - Select *add carry-through*.
35. Add longeron break to assembly 3:
 - Set *X Rotation* to -70.
 - Select *add longeron*.

Before continuing to step 35, your screen should look something like figure C.19.

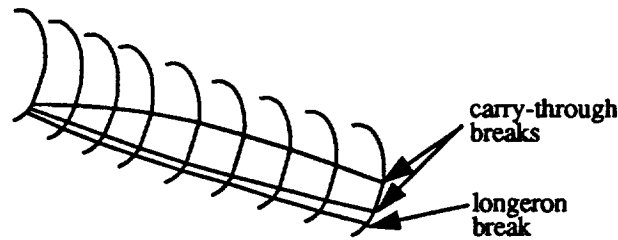


FIGURE C.19. Illustration of screen after step 34 of lesson 1.

36. Add wing-connection and longeron breaks to assembly 4:

- Select *breaks* from the Edit Mode menu.
- Left-click on first and last station of assembly 4 (stations at X = 234 and X = 247).
- Select *add wing-connection*.
- Set *X Rotation* to 56 and select *add longeron*.

Your screen should look like figure C.20.

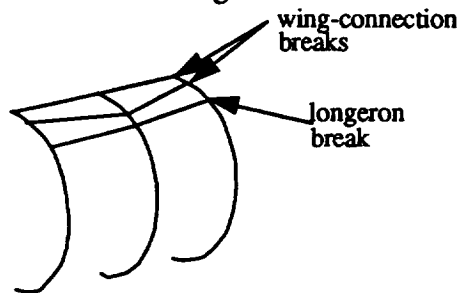


FIGURE C.20. Illustration of screen after step 35 of lesson 1.

37. Add longeron to assemblies 2, 3, and 4.

- Select *breaks* from the Edit Mode menu.
- Left-click on station at X=31 and station at X=247.
- Set *X Rotation* to 0 and select *add longeron*.

38. Add floor and keel beams:

- Set *Z Translation* to -1.
- Select *add keel / floor* to add floor.
- At the prompt, enter patch precision of 7. Patch precision specifies how many rows of patches to fit between the outer and inner edge of the beam.

- Set *X Rotation* to -90.
- Select *add keel / floor* to add keel.
- Enter patch precision of 7.

(See fig. C.21.)

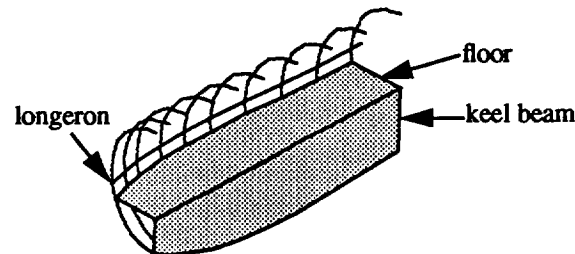


FIGURE C.21.

Step 37 of lesson 1 adds a floor and a keel beam (this figure does not show patch precision).

39. Specify number of patch rows to distribute between breaks (default is 1):

- Select *breaks* from the Edit Mode menu.
- Select all assemblies by right-clicking anywhere in the right View Window.
- Select *patches-between-breaks*.
- Observe the magenta curves that outline the top and bottom of assembly 1. Middle-click. You are prompted to enter the number of patches to distribute in the area between the magenta curves. Enter 8.

Note: The *patches-between-breaks* option is equivalent to distributing equally-spaced longerons. For example, distributing 8 patches in assembly 1 results in the same structural model as if you had added seven equally-spaced longerons. The main difference is that when you add a longeron, right away a you can see where the longeron is added. Unfortunately, you don't get similar visual cues when you use the *patches-between-breaks* option. Until the model is generated, the user is responsible for keeping track of how many patches have been distributed between which breaks.

- Right-click. This moves you to the top of the assembly 2.

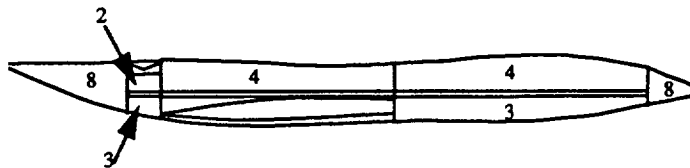


FIGURE C.22.

Diagram showing how many patches to distribute between breaks.

- Using figure C.22 as a guide, distribute longeron between remaining breaks.

This action:	Causes:
left-click	The magenta curves to cycle forward in the assembly. No effect if the assembly has no breaks.
middle-click	The program to query for the number of patch rows to distribute between the two magenta curves.
right-click	Advances the magenta curves to the top of the next assembly. If you are at the last assembly (i.e. assembly 5), right-click ends <i>patches-between-breaks</i> mode.

40. To generate the structural model:

- Select *surfaces* from the Edit Mode Menu.
- Select *fuselage* from the Surfaces Menu.
- Right-click anywhere in the right View Window.
- Before generating carry-through structure for assembly 3, the program queries whether to use the vertical drop algorithm. Select “yes” from the Verify menu.

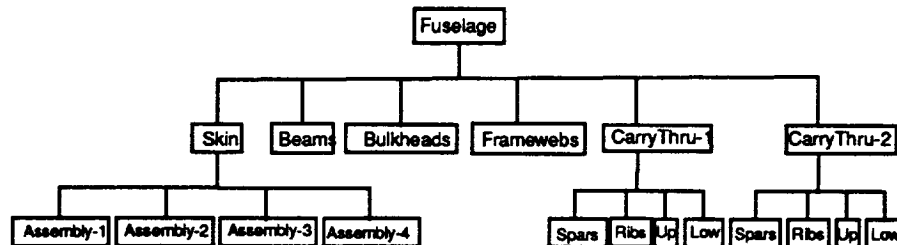


FIGURE C.23.

Hierarchy of components for fuselage of *vehicle1*. Only group nodes are shown in this figure.

Figure C.23 shows the hierarchy of components that are added to the tree. Since the tree is very large, only group nodes are shown in the figure. Each group contains one or more geometric components. Figure C.23 shows the geometry.

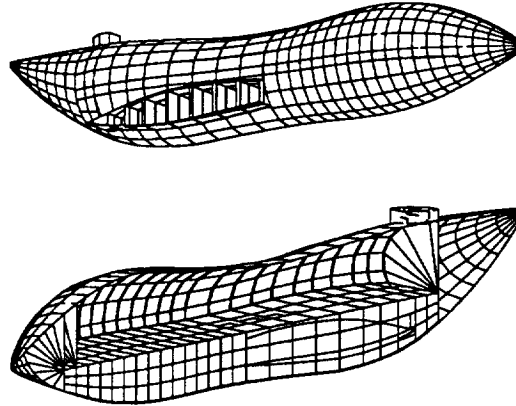


FIGURE C.24. Structural model for fuselage of *vehicle1*.

41. To delete X-Z planar surface from vertical tail's carry-through structure, enter TREE Mode and delete node named *UPPER3 0* from the Tree.

Merge Structural Components

42. Select MERGE from the main menu.
43. Check that the tolerance is set to <.001>.
44. Merge fuselage.
 - Select *merge fuselage* from the Automatic Merge Menu.
 - Select *fuselage* from the tree.
45. Merge wing to fuselage
 - Select *merge wing to fuselage* from the Automatic Merge Menu.
 - Select *mw* from tree.

Lesson 2

Overview

Lesson 2 teaches you how to generate structure that integrates a wing and a tip fin, and how to generate a fuselage model with internal tanks. Figure C.25 diagrams the modeling approach of lesson 2.

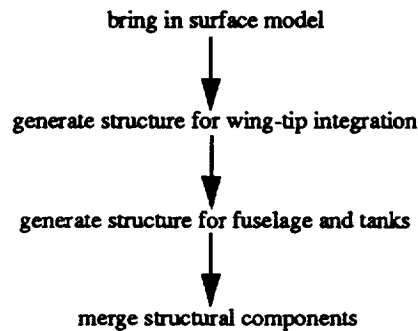


FIGURE C.25. Modeling approach for lesson 2.

Bring in the Surface Model

1. Enter FILE Mode. Read the SMART file that is named *vehicle2*. As shown in figure C.26, the tree that appears in the View Window is organized into two top-level groups of components:
 - Structural models for the wing and tip fin are grouped under the node that is named *Structural*.
 - The remaining components, which are grouped under *External*, define the vehicle's external shape.

Although the wing and tip fin structural models are made up of many components, they each appear in the tree as a single node. These nodes, which appear in the tree as solid blue rectangles, are referred to as compressed groups. Compression is a device that you can use to make large trees more manageable. When you are picking from the tree, you compress a group by doing a shift-left-click on the group's

top-level node. Similarly, shift-left-click on a compressed group expands the group so that all nodes appear in the tree.

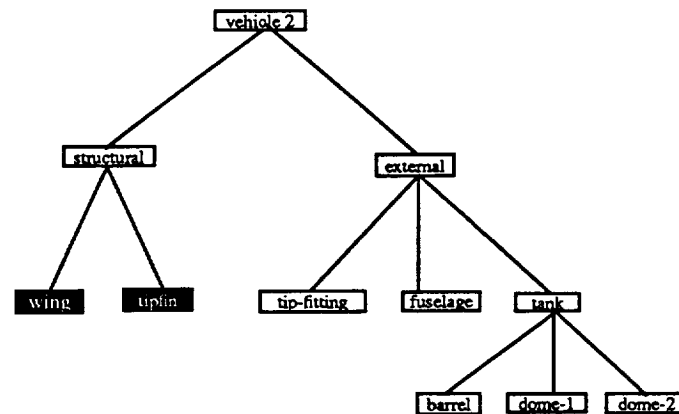


FIGURE C.26. Tree for *vehicle2*. Compressed nodes shown in reverse video.

Generate Structure that Joins Tip Fin and Wing

2. Select S-FUSE from the main menu.
3. Select *tip fitting* from the tree.

S-FUSE was originally designed to accept only reflected surfaces and runs best when this type of input is used. However, in this case a full cylinder (i.e., not reflected) was required for integration of the wing and tip fin.

4. Select tip rib of wing:
 - Shift-left-click on *wing* node. Wing expands to the tree that is shown in figure C.27.
 - Select *mw-btip* from the tree (circled node in fig. C.27.)
 - Select *root rib* from the Component Type menu. This is permissible because both root and tip ribs are processed the same way.
 - SMART extrudes the tip rib away from the tip fitting. To correct, select *no* from the Verify Menu. Enter a distance to extrude.
 Hint: To switch the direction that the root rib is extruded, enter the negative of the current extrusion value.
 - When you are satisfied that both the surfaces intersect the tip fitting (a value of 8 should do it), select yes from the Verify Menu.

Note: Before continuing to step 5, you may want to Shift-left-click on the *wing* node to compress the wing-structural model. Actually, you can compress or ex-

pand any group node in the tree whenever you are prompted to pick from the tree (mouse icon appears in lower right area of screen).

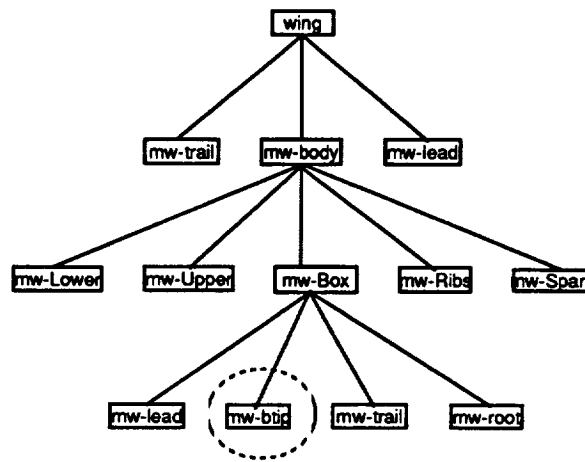


FIGURE C.27. Expanded "wing" node.

5. Select *tip-fin* from tree. SMART extracts the root rib from the structural model. Alternatively, you could have expanded tip-fin and explicitly selected the root rib component. Again, SMART extrudes the rib away from the tip fitting. To correct, select *no* from the Verify Menu and enter a value of 10.
6. Move mouse cursor outside the View Window. (See fig. C.28.)

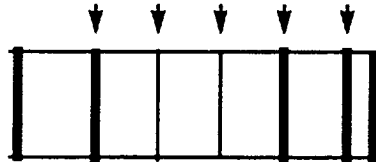


FIGURE C.28. Illustration of screen after step 6 of lesson 2.

7. Delete all stations that are not spars. (See fig. C.29.)

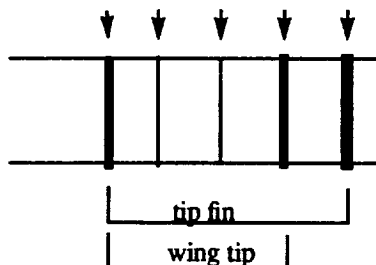


FIGURE C.29. Stations are only at spar locations.

8. Add wing connection breaks.
 - Select *breaks* from the Edit Mode menu.

- Select **ALL** by clicking right mouse button inside View Window.
- Select *add wing-connection* from Stations Menu.
- Notice that breaks for the tip fin are missing from the first assembly. Fix this problem by setting the tolerance to a bigger number,.005 or greater. Select *add wing-connection* from the Breaks menu. Before continuing, check to see that your screen looks like figure C.30.

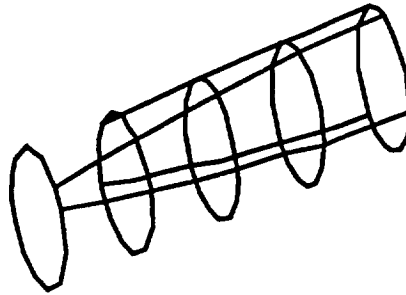


FIGURE C.30. Step 8 of lesson 2 adds wing-connection breaks to the tip-fitting.

9. Add longeron

- Set *Y Translation* to 551 and *X Rotation* to -90.
- Select *add longeron*.

10. Edit second assembly so that breaks are continuous with first assembly.

- Select *breaks* from the Edit Mode menu.
- Middle-click on station at X = 1450. (Note: middle button is useful for when you want to select only one assembly.)
- Small yellow squares show where breaks in the first assembly do not continue into the second assembly. Add a longeron at each marker.

If the yellow square does not disappear, then the longeron that you added is not continuous with the break in the adjoining assembly. In this case, delete the longeron and try again. Use the X-ZM View Option to zoom in on the square. The more you zoom, the easier it is to accurately align the cutting plane.

11. To generate the structural model:

- Select *surfaces* from the Edit Mode Menu and *fuselage* from the Surfaces Menu.
- Move cursor into right View Window and right-click.

The resulting geometry, as shown in figure C.31, is added to the left View Window.

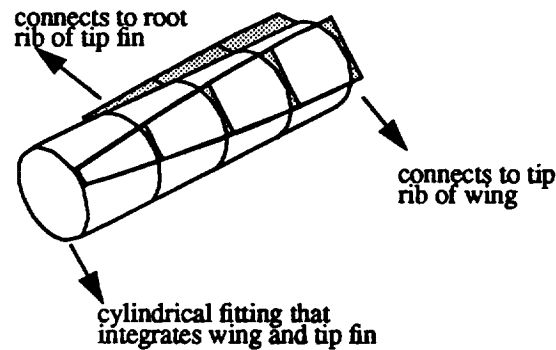


FIGURE C.31. Structural model for tip-fitting of *vehicle2*.

Generate Structural Fuselage with Wing and Internal Tanks

12. Select S-FUSE from the main menu.
13. Select *fuselage* from the tree.
14. Select wing:
 - Select *wing* from tree.
 - Switch the direction that the root rib is extruded by entering the negative of the default extrusion distance.
15. Select tank:
 - Select *dome1* from tree and *dome cap* from Component Type menu.
 - Select *dome2* from tree and *dome cap* from Component Type menu.
 - Select *barrel* from tree and *tank barrel* from Component Type menu.
 - Move mouse cursor out of the View Window.

Note: For best results, pick dome caps from tree before tank barrels.

Before continuing to step 16, your screen should look like figure C.32.

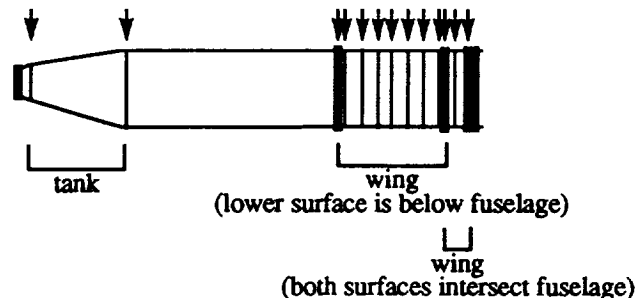


FIGURE C.32. Illustration of screen after step 15 of lesson 2.

16. Distribute stations:

- Select *distribute by count* from the Stations menu.
- Left-click on start station at $X = -497$ and on stop station at $X = -83$.
- The program queries for the number of stations to distribute. Enter 5.
- Repeat, this time distributing 10 stations between $X = -83$ and $X = 859$.

(See fig. 33.)



FIGURE C.33.

Illustration of screen after step 16 of lesson 2.

17. Add wing connection breaks as shown in figure C.34 (not all stations are shown in the figure).

- Select *breaks* from the Edit Mode menu.
- Right-click inside View Window to select ALL stations.
- Select *add wing-connection* from Stations Menu.

Wing breaks are added to two assemblies. There is only one break in the first assembly because the lower wing surface does not intersect the fuselage in this region. The second assembly starts where the lower wing surface curves upward to intersect the fuselage. There are two wing breaks in this assembly because both the upper and the lower wing surfaces intersect this region of the fuselage.

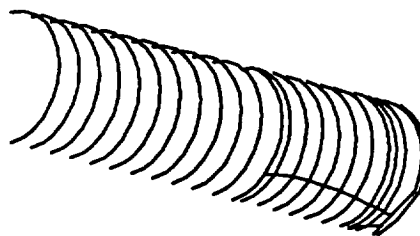


FIGURE C.34.

Illustration of screen after step 17 of lesson 2.

18. Add longerons as shown in figure C.35.

- Set *X Rotation* to 80.
- Select *add longeron*.
- Repeat for *X Rotation* = 70, 60, 50, 40, 30, 20, 10, 0, -10, -20, -30, and -40.

Hint: Use the small arrows on either side of the slider bar to make discrete changes to the slider bar's rotation value. Click on the left arrow to decrement the value or the right arrow to increment. For Rotation slider bars, the right mouse button changes the value 10 degrees, the middle changes the value 1 degree, and the left changes the value <0.1> degree.

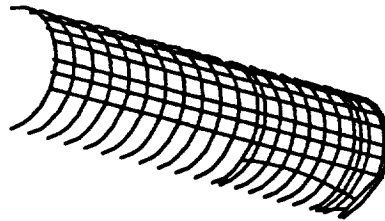


FIGURE C.35. Illustration of screen after step 18 of lesson 2.

19. Add longerons to assemblies in front of the wing.
 - Select *breaks* from the Edit Mode menu.
 - Left-click on start station at X = -546 and stop station at X = 859.
 - Set *X Rotation* to -49.
 - Select *add longeron*.
 - Repeat for *X Rotation* = -60, -70, and -80.
20. Add longeron to second wing assembly.
 - Select *breaks* from the Edit Mode menu.
 - Middle-click on station at X = 1345.
 - Set *X Rotation* to 50 and select *add longeron*.
21. Add longerons to last assembly.
 - Select *breaks* from the Edit Mode menu.
 - Middle-click on station at X = 1450.
 - Add longerons that are continuous with breaks of the adjacent assembly. Refer to step 11 if you are not sure of how this is done.
22. To generate the structural model:
 - Select *surfaces* from the Edit Mode Menu.
 - To generate the fuselage: select *fuselage* from the Surfaces menu, and right-click inside the right View Window.
 - Select *internal tanks* from the Surfaces menu to generate the tank.

The resulting geometry, as shown in figure C.36, is drawn in the left View Window.

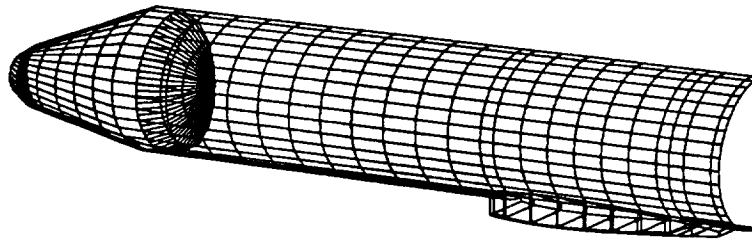


FIGURE C.36. Structural model for fuselage of *vehicle2*.

Merge Structural Components

23. Examine the structural models of the fuselage, tip fitting and wings. As shown in figure C.37, the one problem area in the model is where the wing intersects the fuselage. To fix:

- Select **MERGE** from the main menu.
- Select *merge fuselage* from the Automatic Merge menu.
- Pick top node of tree that was created for the fuselage model.

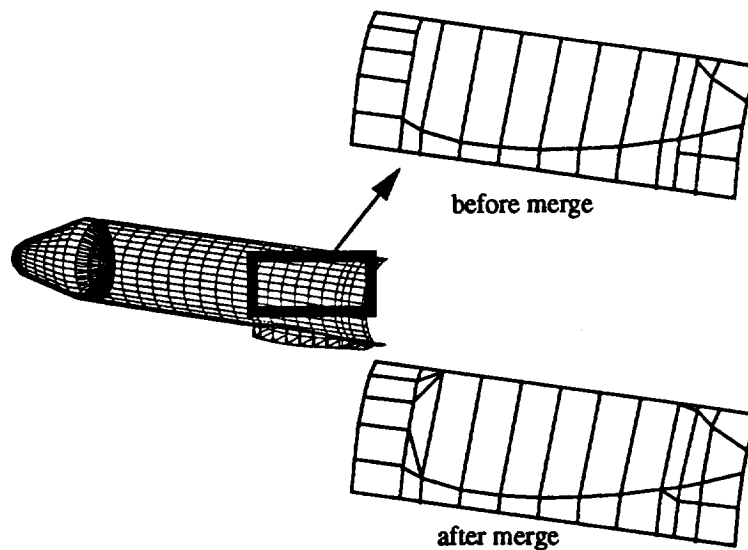


FIGURE C.37. The *merge fuselage* Option is used to correct problem area where wing intersects the fuselage.

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE June 1996		3. REPORT TYPE AND DATES COVERED Contractor Report
4. TITLE AND SUBTITLE SMART Structures User's Guide - Version 3.0			5. FUNDING NUMBERS C NAS1-20255 WU 242-30-03-40	
6. AUTHOR(S) Jan L. Spangler				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Analytical Services and Materials, Inc. Hampton, VA 23666			8. PERFORMING ORGANIZATION REPORT NUMBER	
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Langley Research Center Hampton, VA 23681-0001			10. SPONSORING / MONITORING AGENCY REPORT NUMBER NASA CR-201582	
11. SUPPLEMENTARY NOTES Langley Technical Monitor: Lawrence F. Rowell Final Report				
12a. DISTRIBUTION / AVAILABILITY STATEMENT Unclassified - Unlimited Subject Category - 61			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) Version 3.0 of the Solid Modeling Aerospace Research Tool (SMART Structures) is used to generate structural models for conceptual and preliminary-level aerospace designs. Features include the generation of structural elements for wings and fuselages, the integration of wing and fuselage structural assemblies, and the integration of fuselage and tail structural assemblies. The highly interactive nature of this software allows the structural engineer to move quickly from a geometry that defines a vehicle's external shape to one that has both external components and internal components which may include ribs, spars, longerons, variable depth ringframes, a floor, a keel, and fuel tanks. The geometry that is output is consistent with FEA requirements and includes integrated wing and empennage carry-through and frame attachments. This report provides a comprehensive description of SMART Structures and how to use it.				
14. SUBJECT TERMS Mechanical CAD; Surface generation; Structural design; Vehicle design; Three-dimensional geometry modeling			15. NUMBER OF PAGES 81	
			16. PRICE CODE A05	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT	20. LIMITATION OF ABSTRACT	

